

MIKE SHE USER MANUAL

VOLUME 2: REFERENCE GUIDE



8 December 2006 1:28 pm



Please Note

Copyright

This document refers to proprietary computer software which is protected by copyright. All rights are reserved. Copying or other reproduction of this manual or the related programs is prohibited without prior written consent of DHI Water & Environment (DHI). For details please refer to your 'DHI Software Licence Agreement'.

Limited Liability

The liability of DHI is limited as specified in Section III of your 'DHI Software Licence Agreement':

'IN NO EVENT SHALL DHI OR ITS REPRESENTATIVES (AGENTS AND SUPPLIERS) BE LIABLE FOR ANY DAMAGES WHATSOEVER INCLUDING, WITHOUT LIMITATION, SPECIAL, INDIRECT, INCIDENTAL OR CONSEQUENTIAL DAMAGES OR DAMAGES FOR LOSS OF BUSINESS PROFITS OR SAVINGS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION OR OTHER PECUNIARY LOSS ARISING OUT OF THE USE OF OR THE INABILITY TO USE THIS DHI SOFTWARE PRODUCT, EVEN IF DHI HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. THIS LIMITATION SHALL APPLY TO CLAIMS OF PERSONAL INJURY TO THE EXTENT PERMITTED BY LAW. SOME COUNTRIES OR STATES DO NOT ALLOW THE EXCLUSION OR LIMITATION OF LIABILITY FOR CONSEQUENTIAL, SPECIAL, INDIRECT, INCIDENTAL DAMAGES AND, ACCORDINGLY, SOME PORTIONS OF THESE LIMITATIONS MAY NOT APPLY TO YOU. BY YOUR OPENING OF THIS SEALED PACKAGE OR INSTALLING OR USING THE SOFTWARE, YOU HAVE ACCEPTED THAT THE ABOVE LIMITATIONS OR THE MAXIMUM LEGALLY APPLICABLE SUBSET OF THESE LIMITATIONS APPLY TO YOUR PURCHASE OF THIS SOFTWARE.'

Printing History

December 2007
Edition 2007





Phi Software

CONTENTS



The MIKE SHE Reference Guide	15
1 REFERENCE GUIDE OVERVIEW	17
2 SETUP DATA TAB	19
2.1 Display	20
2.1.1 Foreground/Background	21
2.1.2 Image Overlays	22
2.1.3 Grid Overlays	23
2.1.4 Shape Overlays	24
2.1.5 River Overlays	25
2.1.6 MIKE SHE Well Database Overlays	25
2.1.7 Current Layer	26
2.2 Simulation Specification	26
2.2.1 Simulation Title	28
2.2.2 Simulation Period	28
2.2.3 Time Step Control	30
2.2.4 OL Computational Control Parameters	34
2.2.5 UZ Computational Control Parameters	39
2.2.6 SZ Computational Control Parameters	40
2.3 Water Quality Simulation Specification	45
2.3.1 WQ Simulation Title	46
2.3.2 WQ Simulation Period	47
2.3.3 Water Quality Time Step Control	49
2.4 Species	50
2.5 Species Parameters	51
2.6 Model Domain and Grid	52
2.7 Subcatchments	54
2.7.1 River Links	55
2.8 Topography	56
2.9 Precipitation	57
2.9.1 Precipitation Rate	58
2.9.2 Net Rainfall Fraction	59
2.9.3 Infiltration Fraction	59
2.9.4 Air Temperature	60
2.9.5 Snowmelt Constants	62
2.10 Land Use	62
2.10.1 Vegetation	64
2.10.2 Paved Runoff Coefficient	68
2.10.3 Irrigation Command Areas	69
2.10.4 Sheet Application Area	76
2.10.5 Irrigation Demand	76
2.10.6 Irrigation Priorities	78



2.11	Evapotranspiration	79
2.11.1	Reference Evapotranspiration	79
2.12	Rivers and Lakes	80
2.12.1	Flood codes	81
2.12.2	Bathymetry	82
2.13	Overland Flow	82
2.13.1	Manning number	83
2.13.2	Detention Storage	84
2.13.3	Initial Water Depth	85
2.13.4	Overland-groundwater Leakage Coefficient	85
2.13.5	Separated Flow Areas	87
2.13.6	Overland Flow Zones	87
2.13.7	Dispersion coefficient along columns/rows	89
2.13.8	Initial Mass (per Species)	89
2.14	Unsaturated Zone	90
2.14.1	Soil Profile Definitions	93
2.14.2	Groundwater Depths used for UZ Classification	96
2.14.3	Ground Water Table	97
2.14.4	Partial automatic classification	97
2.14.5	Specified classification	98
2.14.6	2-Layer UZ soil properties	99
2.14.7	ET Surface Depth	100
2.15	Groundwater Table	101
2.16	Saturated Zone	102
2.16.1	Interflow Reservoirs	105
2.16.2	Baseflow Reservoirs	106
2.16.3	Geological Units	108
2.16.4	Geological Layers	109
2.16.5	Geological Lenses	110
2.16.6	Lower Level (Geological Layer or Lense, or Water Quality Layer)	111
2.16.7	Upper Level	111
2.16.8	Horizontal Extent	111
2.16.9	Geological Unit Distribution	112
2.16.10	Horizontal Hydraulic Conductivity	112
2.16.11	Vertical Hydraulic Conductivity	113
2.16.12	Specific Yield	114
2.16.13	Specific Storage	114
2.16.14	Porosity	115
2.16.15	Dispersion Coefficients LHH, THH, TVH, LVV, THV	115
2.16.16	Computational Layers	116
2.16.17	Lower Level (Numerical Layer)	117
2.16.18	Initial Potential Head	117



2.16.19	Outer boundary conditions	118
2.16.20	Internal boundary conditions	121
2.16.21	Initial concentration	123
2.16.22	Drainage	123
2.16.23	Drain Level	125
2.16.24	Drain Time Constant	126
2.16.25	Drain Codes	127
2.16.26	Option Distribution	128
2.16.27	Pumping Wells	129
2.17	Sources	130
2.17.1	Extent	132
2.17.2	Top Elevation	133
2.17.3	Bottom Elevation	133
2.17.4	Strength	134
2.18	Storing of Results	135
2.18.1	Detailed time series output	138
2.18.2	Detailed MIKE 11 Output	141
2.18.3	Grid series output	143
2.19	Extra Parameters	144
3	PREPROCESSED DATA TAB	145
3.1	Processed Data	146
3.1.1	Model Domain and Grid	147
3.1.2	Precipitation and Evapotranspiration	148
3.1.3	River Links	148
3.1.4	UZ Soil Profile Grid Codes	149
3.1.5	UZ Classification Grid Codes	150
3.1.6	Saturated Zone Items	151
3.1.7	Saturated Zone Drainage	151
3.2	GeoScene3D	152
4	RESULTS TAB	155
4.1	MIKE SHE Detailed Time Series	156
4.2	Gridded Data Results Viewer	157
4.2.1	“The Result Viewer setup file already exists” warning	158
4.3	MIKE 11 Detailed Time Series	159
4.4	Run Statistics	160
4.4.1	Shape file output for run statistics	161
4.4.2	Statistic Calculations	161
4.5	GeoScene3D	163
	MIKE SHE Editors	165



5	WELL EDITOR	167
	5.0.1 Interactive Map	167
	5.0.2 Well Locations	168
	5.0.3 Well Filters	169
	5.0.4 Layers Display	170
	5.1 Importing Data	170
	5.1.1 Importing data from a .t0 file	171
	5.1.2 Importing TAB delimited text file	172
6	UZ SOIL PROPERTIES EDITOR	173
	6.1 Soil Moisture Retention Curve	173
	6.1.1 Tabulated	174
	6.1.2 Van Genuchten and Campbell Functions	174
	6.2 Hydraulic Conductivity Function	174
	6.2.1 Tabulated	175
	6.2.2 Averjanov	175
	6.2.3 Van Genuchten and Campbell/Burdine methods	176
7	ET VEGETATION PROPERTIES EDITOR	177
	7.1 Vegetation Database Items	177
	7.1.1 Specifying Vegetation Properties in a Database	177
	7.1.2 Vegetation stages	178
	7.1.3 Evapotranspiration Parameters	179
	7.1.4 Vegetation Development Table	180
	7.1.5 Irrigation Parameters	181
	7.2 Example database	183
8	WATER BALANCE EDITOR	185
	8.1 Extraction	186
	8.2 Postprocessings	187
	8.2.1 Postprocessing Detail	188
	8.3 Results	190
9	PARTICLE TRACKING EDITOR	193
	9.1 Running MODPATH outside of MIKE SHE	194
	9.2 Using MODPATH in the MIKE SHE GUI	195
	9.3 Setup Tab	196
	9.3.1 Display	196
	9.3.2 Flow model setup	196
	9.4 Simulation Tab	197
	9.4.1 Transport Simulations	197
	9.4.2 MODPATH simulation specification	198
	9.4.3 Particles	199
	9.5 Results Tab	200



10	SIMPLE SHAPE EDITOR	201
11	MIKE SHE TOOLBOX	203
11.1	Data Analysis	203
11.1.1	Grid series file compare	203
11.2	File Converter	204
11.2.1	dfs2+dfs0 to dfs2	204
11.2.2	t0 to dfs0 and t2 to dfs2	205
11.3	Util	205
11.3.1	Grid calculator	205
	Technical Reference for Water Movement	207
12	WATER MOVEMENT OVERVIEW	209
13	OVERLAND FLOW - REFERENCE	211
13.1	Finite Difference Method	211
13.1.1	Diffusive Wave Approximation	211
13.1.2	Finite Difference Formulation	214
13.1.3	Successive Over-Relaxation (SOR) Numerical Solution	216
13.1.4	Explicit Numerical Solution	217
13.1.5	Boundary conditions	218
13.1.6	Low gradient damping function	218
13.2	Simplified Overland Flow Routing	220
13.2.1	Theoretical basis	220
13.2.2	Implementation in MIKE SHE	223
13.2.3	Coupling to other processes	223
13.2.4	Avoiding the redistribution of ponded water	224
13.2.5	Routing simple overland flow directly to the river	225
14	CHANNEL FLOW - REFERENCE	227
14.1	Coupling of MIKE SHE and MIKE 11	228
14.1.1	MIKE SHE Branches vs. MIKE 11 Branches	230
14.1.2	The River-Link Cross-section	231
14.1.3	Connecting MIKE 11 Water Levels and Flows to MIKE SHE	232
14.2	River-Aquifer Exchange (line source/link)	234
14.2.1	Aquifer Only Conductance	234
14.2.2	River bed only conductance	235
14.2.3	Both aquifer and river bed conductance	237
14.2.4	Steady-state groundwater simulations	238
14.3	Area Inundation using Flood Codes (areal source/sink)	239
14.3.1	Determination of the Flooded Area and Water Levels	239
14.3.2	Calculation of the Exchange Flows	239
14.4	Direct Overbank Spilling to and from MIKE 11	240



15	EVAPOTRANSPIRATION - REFERENCE	243
15.1	Kristensen and Jensen method	244
15.1.1	Sublimation from Snow	244
15.1.2	Canopy Interception	244
15.1.3	Evaporation from the Canopy	245
15.1.4	Plant Transpiration	246
15.1.5	Soil Evaporation	250
15.1.6	Evapotranspiration Coefficients C1, C2 and C3	251
15.2	Simplified ET for the Two-Layer Water Balance Method	254
15.2.1	Sublimation from Snow	255
15.2.2	Canopy Interception	255
15.2.3	Soil Moisture	256
15.2.4	Infiltration	258
15.2.5	Evapotranspiration	258
15.2.6	Recharge to the Saturated Zone	260
16	UNSATURATED FLOW - REFERENCE	261
16.1	Richards Equation	262
16.1.1	Numerical Solution	263
16.1.2	Boundary Conditions	268
16.1.3	Initial Conditions	271
16.1.4	Sources and sinks	272
16.1.5	Spatial resolution	272
16.2	Gravity Flow	273
16.2.1	Solution method	274
16.2.2	Initial Conditions	275
16.3	Two-Layer Water Balance	275
16.3.1	Soil Moisture	276
16.3.2	Infiltration	279
16.4	Simplified Macropore Flow (bypass flow)	280
16.5	Lumped UZ Calculations	281
16.6	Coupling the Unsaturated Zone to the Saturated Zone	282
16.6.1	Steps in the Coupling Procedure	284
16.6.2	Limitations	287
16.6.3	Evaluation of the UZ-SZ Coupling	287
17	SATURATED FLOW - REFERENCE	289
17.1	3D Finite Difference Method	289
17.1.1	The Pre-conditioned Conjugate Gradient (PCG) Solver	290
17.1.2	PCG Steady State Solver	294
17.1.3	Boundary Conditions	296
17.1.4	Initial Conditions	300
17.1.5	The Sheet Pile Module	300



17.1.6	The Successive Overrelaxation (SOR) Solver	301
17.2	Linear Reservoir Method	307
17.2.1	Linear Reservoir Theory	308
17.2.2	General Description	309
17.2.3	Subcatchments and Linear Reservoirs	312
17.2.4	Calculation of Interflow	313
17.2.5	Calculation of Interflow Percolation and Dead Zone Storage	315
17.2.6	Calculation of Baseflow	316
17.2.7	UZ Coupling	317
17.2.8	Coupling to Mike 11	318
17.2.9	Limitations of the Linear Reservoir Method	318
Technical Reference for Water Quality		321
18	WATER QUALITY OVERVIEW	323
19	ADVECTION DISPERSION - REFERENCE	325
19.1	Simulation control	325
19.1.1	Flow Storing Requirements	325
19.1.2	Internal Boundary Conditions	325
19.1.3	Time step calculations	327
19.2	Solute Transport in the Saturated Zone	329
19.2.1	Governing Equations	330
19.2.2	Solution Scheme	334
19.2.3	Initial Conditions	338
19.2.4	Source/Sinks, Boundary Conditions and other Exchanges	338
19.2.5	Transport in Fractured Media	339
19.3	Solute Transport in the Unsaturated Zone	340
19.3.1	Governing Equations	340
19.3.2	Solution Scheme	341
19.3.3	Initial Conditions	343
19.3.4	Source/Sinks, Boundary Conditions and other Exchanges	343
19.4	Solute Transport in Overland Flow	344
19.4.1	Governing Equations	345
19.4.2	Solution scheme	345
19.4.3	Initial Conditions	347
19.4.4	Source/Sinks, Boundary Conditions and other Exchanges	348
19.5	Solute Transport in MIKE 11	348
20	REACTIVE TRANSPORT - REFERENCE	351
20.1	Sorption	351
20.1.1	Equilibrium Sorption Isotherms	352
20.1.2	Kinetic Sorption Isotherms	355
20.1.3	Sorption in Dual Porosity Systems	355



20.2	Decay	356
20.3	Plant Uptake	359
20.4	Process verification	360
	20.4.1 Other Processes - Simulation Examples	366
21	PARTICLE TRACKING-REFERENCE	369
	21.1 Governing equations	369
References		375
Index		381





Phi Software

THE MIKE SHE REFERENCE GUIDE





1 **REFERENCE GUIDE OVERVIEW**

The Reference Guide includes detailed descriptions of the tools and dialogues that you are likely to encounter as you develop a MIKE SHE model. If you click F1 in any MIKE SHE dialogue, you will land in one of the sections of this manual. Likewise, if you click F1 in any MIKE 11 or other MIKE Zero dialogue, you will land in an appropriate section of the on-line help.

The first part of the document follows the general organization of the MIKE SHE user interface. These sections are the places you land, when you press the F1 key in MIKE SHE and include

- the Setup Data Tab (*V.2 p. 19*)
- the Preprocessed Data Tab (*V.2 p. 145*), and
- the Results Tab (*V.2 p. 155*)

This is followed by detailed descriptions of the various MIKE SHE specific editors, including

- the Well editor (*V.2 p. 167*)
- the UZ Soil Properties Editor (*V.2 p. 173*)
- the ET Vegetation Properties Editor (*V.2 p. 177*)
- the Water Balance Editor (*V.2 p. 185*)
- the Particle Tracking Editor (*V.2 p. 193*), and
- the Simple Shape Editor (*V.2 p. 201*)

In turn, this is followed by the MIKE SHE Technical Reference. For the Water Movement module, this includes

- the Overland Flow - Reference (*V.2 p. 211*),
- the Channel Flow - Reference (*V.2 p. 227*),
- the Evapotranspiration - Reference (*V.2 p. 243*),
- the Unsaturated Flow - Reference (*V.2 p. 261*), and
- the Saturated Flow - Reference (*V.2 p. 289*).

For the Water Quality module, this includes

- the Advection Dispersion - Reference (*V.2 p. 325*),
- the Reactive Transport - Reference (*V.2 p. 351*), and



- the Particle Tracking-Reference (*V.2 p. 369*).



2 **SETUP DATA TAB**

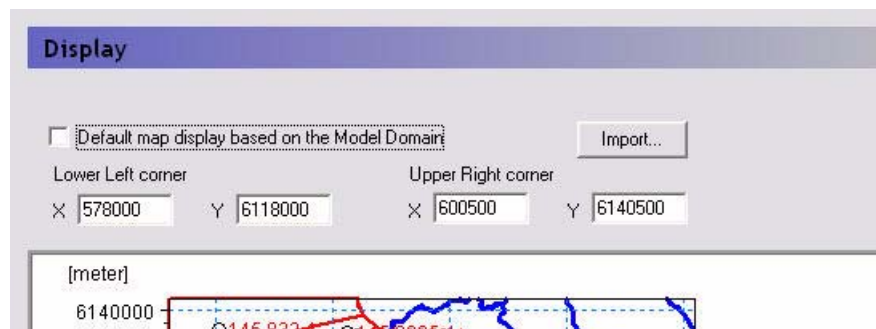
This chapter is organized around the Setup Data Tree. For each branch in the data tree there is a corresponding subsection. The main sections include:

- **Display (V.2 p. 20)** - display of map overlays
- **Simulation Specification (V.2 p. 26)** - control and selection of water movement engines
- **Water Quality Simulation Specification (V.2 p. 45)** - control and selection of water quality engines
- **Species (V.2 p. 50)** - specification of species for water quality simulations
- **Model Domain and Grid (V.2 p. 52)** - definition of model extent and grid
- **Subcatchments (V.2 p. 54)** - definition of catchment boundaries for lumped parameter water movement engines
- **Topography (V.2 p. 56)** - specification of land surface elevation
- **Precipitation (V.2 p. 57)** - specification and extent of precipitation measurements
- **Land Use (V.2 p. 62)** - specification of vegetation and irrigation
- **Evapotranspiration (V.2 p. 79)** - specification and extent of reference evapotranspiration measurements or calculations
- **Rivers and Lakes (V.2 p. 80)** - link to MIKE 11 channel flow model
- **Overland Flow (V.2 p. 82)** - specification of 2D overland sheet flow parameters for both water movement and water quality
- **Unsaturated Zone (V.2 p. 90)** - specification of 1D unsaturated zone columns
- **Groundwater Table (V.2 p. 101)** - specification of static lower boundary condition for unsaturated flow, if saturated zone not included
- **Saturated Zone (V.2 p. 102)** - specification of 3D saturated zone parameters for both water movement and water quality
- **Sources (V.2 p. 130)** - location and extent of solute sources for water quality simulation
- **Storing of Results (V.2 p. 135)** - output selection for calibration time series and gridded data



- **Extra Parameters (V.2 p. 144)** - extra input data for model options not yet available in the data tree

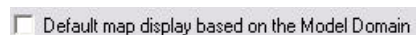
2.1 Display



From this dialogue and data tree branch, you can control the map overlays and size of the map view in the rest of the dialogues.

In any map view in the Setup Data tab, you can right click and chose Zoom In from the pop-up menu. The enlarged view of your map is persistent across all of the map views in the Setup Data tab, as well as to the Processed Data tab.

Also in the right click pop-up menu is a Zoom Extents function, which zooms the map view out to the full extents. By default, the maximum extents of the map view in the MIKE SHE dialogues is set to the size of the model, as defined in the Model Domain and Grid dialogue. However, un-checking the checkbox,



you can chose to set the lower left and upper right coordinates of the maximum extents of the map view.

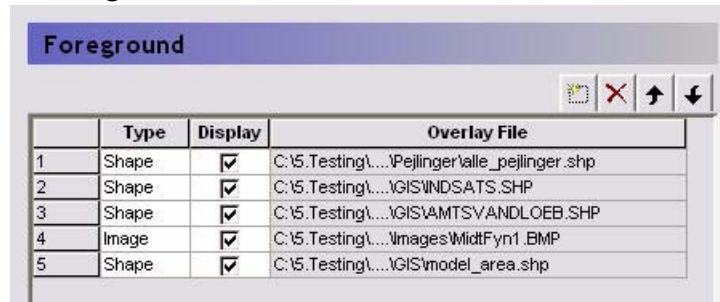
Import - The Import button allows you to read the coordinate extents from a map file, such as a .dfs2 file or a .shp file.

Related Items:

- **Model Domain and Grid (V.2 p. 52)**



2.1.1 Foreground/Background



The Foreground and Background items are used to add map overlays to the map view. The table gives you an overview of the defined overlays and allows you to add, delete and hide overlays.

The order of the overlays in the list controls to some extent the way the overlays are displayed. Furthermore, the Foreground/Background choice determines whether the overlays are displayed in front of or behind the current grid, which in turn controls the way the colours etc. are displayed. The best way to understand the way the overlays are displayed, is to simply play around with a model and some maps to see how the display changes when the map is placed in the foreground vs background, or the order is changed.

The available map types includes,

- ESRI shape (.shp) files,
- Grid (.dfs2) files,
- Image (.bmp, .gif and .jpg) files,
- MIKE 11 river (.nwk11) files, and
- MIKE SHE well database (.wel) files.



2.1.2 Image Overlays

Image Overlay

☒ Display

Image File: C:\5.Testing\NRSøby\Images\MidtFyn1.BMP

Area Coordinates

	x:	y:
Min coords:	575845.9912 [m]	6117918.976 [m]
Max coords:	600867.9912 [m]	6138782.001 [m]

Import geo reference from file...

Image Styles

Display style: Blend colors Transparent color: []

If you want to display a background image in your map view, then you should add an Image overlay. The available image formats include: .bmp, .gif and .jpg.

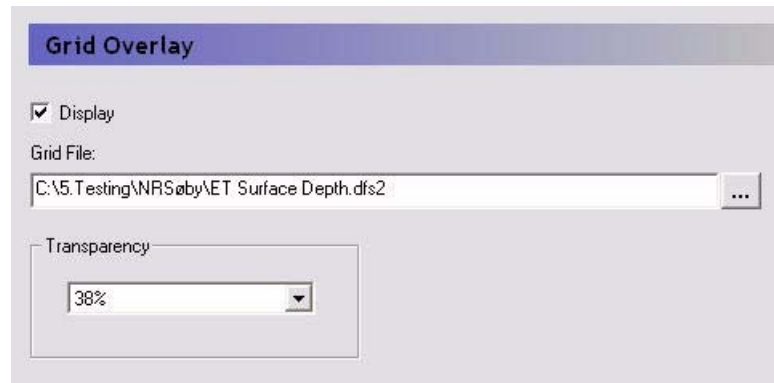
Area Coordinates - Since image files do not contain geographic information, you must specify the spatial location of the image. This is done by specifying the map coordinates of the lower left corner of the image (Minimum X and Y) and the upper right corner of the image (Maximum X and Y).

Import geo reference from file... - Some DHI programs allow you to geo reference an image file, in which case either a .bpw or a .bpmw file is created that contains the origin coordinates and some scaling information.

Image Styles - The Image Style is related to the way the pixel colours are averaged (or not averaged) in overlying grids and images. The Image Style variables have little influence on the image display when the image is displayed in the Background. However, in the Foreground, the Image style is very important. The best results are obtained with the Blend Colours selection., in which case the Transparent colour option is not used.



2.1.3 Grid Overlays



If you want to add a static .dfs2 grid on the map view, then you should add a Grid Overlay. If you add a time varying .dfs2 file the program will not object, but only the first time value will be displayed. If you add a .dfs2 file containing multiple items, such as a results file, you can select the grid item to display from within the file browser.

Transparency - This is used to control the way pixel colours are averaged for displaying images that overlay one another.



2.1.4 Shape Overlays

If you want to add an ESRI shape (.shp) file to the map view, then you must select a Shape overlay.

Shape File and Item - In the file browser dialogue you can select from the available items in the .shp file.

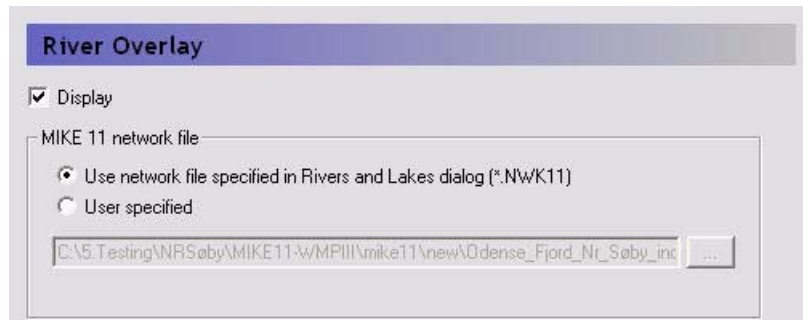
Parameters for Points - This section allows you to customize the way point .shp themes are displayed.

Parameters for Lines and Polygons - This section allows you to customize the way line and polygon themes are displayed.

Units - The ESRI .shp format does not include information on whether the length units are SI or Imperial. So, this combobox allows you to select the length units from a range of SI and Imperial length units.



2.1.5 River Overlays

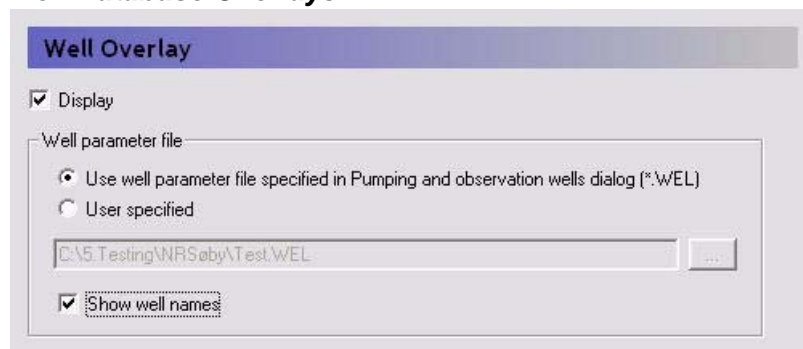


If you want to display a MIKE 11 River network in your map views, then you can add a River Overlay. By default, the river network defined in the Rivers and Lakes dialogue is displayed. If you would rather display a different river network, for example, an overview network with fewer branches, then you can chose **User Specified** and specify the river network file.

Related Items:

- Rivers and Lakes (V.2 p. 80)

2.1.6 MIKE SHE Well Database Overlays



If you want to display a MIKE SHE Well database in your map views, then you can add a Well Overlay. By default, the well database defined in the Pumping Wells dialogue is displayed. If you would rather display a different well database, for example, an overview database with fewer wells, then you can chose **User Specified** and specify the well database file.

Show well names This checkbox turns the well names on and off in the map view.

Related Items:

- Pumping Wells (V.2 p. 129)



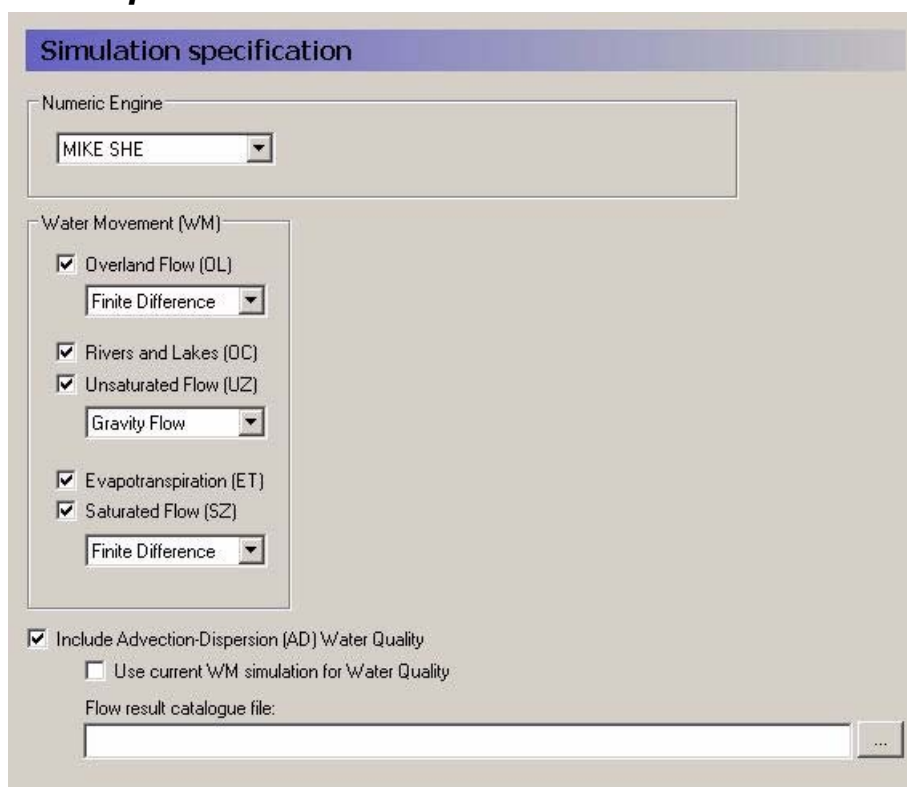
2.1.7 Current Layer



The Current Layer item refers to the grid item currently being displayed in the current map view.

Transparency - This is used to control the way pixel colours are averaged for displaying images that overlay one another.

2.2 Simulation Specification



The Simulation Specification dialogue is the key dialogue in the program. In this dialogue, you can select the key options for each of the components included in the simulation, including:

- Overland Flow (see also Overland Flow - Reference (V.2 p. 211)),



- the Finite Difference Method (V.2 p. 211), or
- a Simplified Overland Flow Routing (V.2 p. 220) method
- Rivers and Lakes (see also Channel Flow - Reference (V.2 p. 227)),
- Unsaturated Flow (see also Evapotranspiration - Reference (V.2 p. 243)),
 - a 1D Richards Equation (V.2 p. 262) solution,
 - a simplified 1D Gravity Flow (V.2 p. 273) solution, or
 - a Two-Layer Water Balance (V.2 p. 275) solution for shallow water tables.
- Evapotranspiration (see also Unsaturated Flow - Reference (V.2 p. 261)) and
- Saturated Flow (see also Saturated Flow - Reference (V.2 p. 289))
 - 3D Finite Difference Method (V.2 p. 289), or
 - a Linear Reservoir Method (V.2 p. 307).

These choices are immediately reflected in the data tree, where the appropriate parameters are added or removed.

There is only one calculation option in this dialogue for Rivers and Lakes because the calculation methods are defined in the MIKE 11 User Interface. Likewise, the use of the simple or advanced Evapotranspiration methods are defined by the unsaturated flow method selected.

Water Quality options

Include Advection Dispersion (AD) Water Quality

At the bottom of this dialogue is a checkbox, where you can specify whether or not to include water quality in the simulation. If checked, the data tree will expand to include water quality data items.

Use Current WM simulation for Water Quality

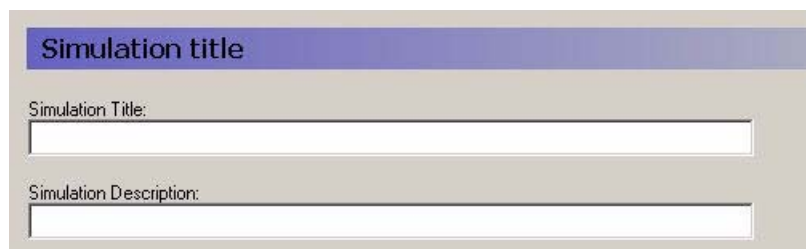
If you uncheck this check box, then you will be able to specify a different different water movement simulation as the source of the cell-by-cell flows for the water quality simulation. This allows you to use one water quality setup and calculate water quality based on several water movement scenarios. You must be careful though to not overwrite your results files from the previous water quality simulations.

For more information on the Advection Dispersion Water Quality see



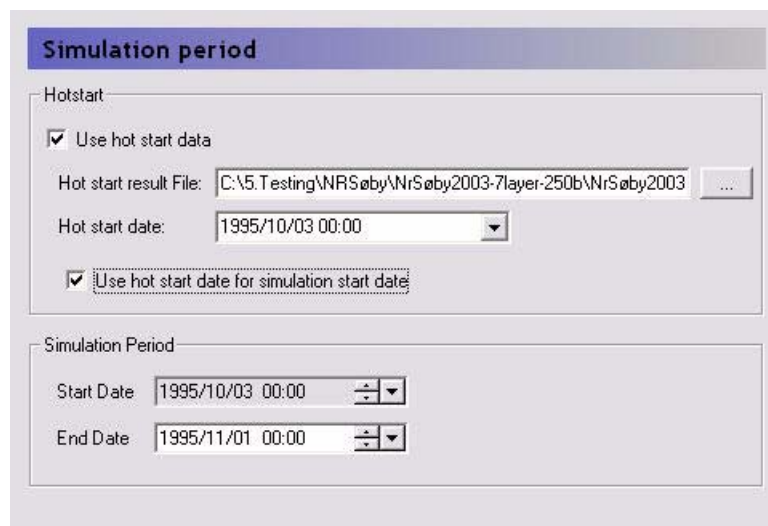
- Simulating Water Quality (V.1 p. 193)
- Advection Dispersion - Reference (V.2 p. 325)

2.2.1 Simulation Title



Title and Description - The Title and Description will be written to output files and appear on plots of the simulation results.

2.2.2 Simulation Period



In the MIKE SHE GUI, all of the simulation output is in terms of real dates, which makes it easy to coordinate the input data (e.g. pumping rates), the simulation results (e.g. calculated heads) and field observations (e.g. measured water levels).

The Simulation Period dialogue is primarily used to define the beginning and end of a transient simulation or the beginning and end of the averaging period for a steady state simulation.

However, the simulation can be started from a hot start file. A hot start file is useful for simulations requiring a long warm up period or for generating initial conditions for scenario analysis. To start a model from a previous



model run, you must first save the hot start data, in the **Storing of Results (V.2 p. 135)** dialogue.

Hot start date - The hot start information is saved at specified intervals and the list of hot start dates is automatically filled in from the hot start file.

Use Hot start date for simulation start date - if you select this option, the simulation start date is greyed out and the simulation starts from the selected hot start date. Otherwise, you are free to chose an independent starting date and only the hot start data is simply used as initial conditions.

Related Items:

- **Storing of Results (V.2 p. 135)**



2.2.3 Time Step Control

Time step control

Time Steps

Initial time step

6

[hrs]

Max allowed OL time step

0.5

[hrs]

Max allowed UZ time step

2

[hrs]

Max allowed SZ time step

24

[hrs]

Increment of reduced time step length

Increment rate (0-1)

0.05

Parameters for Precipitation-dependent time step control

Max precipitation depth per time step

10

[mm]

Max infiltration amount per time step

10

[mm]

Input precipitation rate requiring its own time step

0.1

[mm/hr]

Time Step Control

Conditions: Maximum OL, UZ, and SZ time steps are only active when the component is selected

Variable	Dimensions
Initial time step	[hrs]
Max allowed OL time step	[hrs]
Max allowed UZ time step	[hrs]
Max allowed SZ time step	[hrs]
Increment rate	-
Max precipitation depth per time step	[mm]
Max infiltration amount per time step	[mm]
Input precipitation rate requiring its own time step	[mm/hr]

Time Steps

Initial time step This is the initial time step for all of the components, unless the component's maximum allowed time step is less than the initial time step.



Max allowed time steps Each of the main hydrologic components in MIKE SHE run with independent time steps. Although, the time step control is automatically controlled, whenever possible, MIKE SHE will run with the maximum allowed time steps.

Note In the 2007 Release the MIKE 11 time step is no longer specified in MIKE SHE.

The component time steps are independent, but they must meet to exchange flows, which leads to some restrictions on the specification of the maximum allowed time steps.

- If MIKE 11 is running with a constant time step, then the Max allowed Overland (OL) time step must be a multiple of the MIKE 11 constant time step. If MIKE 11 is running with a variable time step, then the actual OL time step will be truncated to match up with the nearest MIKE 11 time step.
- The Max allowed UZ time step must be an even multiple of the Max allowed OL time step, and
- The Max allowed SZ time step must be an even multiple of the Max allowed UZ time step.

Thus, the overland time step is always less than or equal to the UZ time step and the UZ time step is always less than or equal to the SZ time step.

If you are using the implicit solver for overland flow, then a maximum OL time step equal to the UZ time step often works. However, if you are using the explicit solver for overland flow, then a much smaller maximum time step is necessary, such as the default value of 0.5 hours.

If the unsaturated zone is included in your simulation and you are using the Richards equation or Gravity Flow methods, then the maximum UZ time step is typically around 2 hours. Otherwise, a maximum time step equal to the SZ time step often works.

Groundwater levels react much slower than the other flow components. So, a maximum SZ time step of 24 or 48 hours is typical, unless your model is a local-scale model with rapid groundwater-surface water reactions.

Increment of reduced time step length

Increment rate This is a factor for both decreasing the time step length and increasing the time step length back up to the maximum time step, after the time step has been reduced. See Parameters for Precipitation-



dependent time step control (V.2 p. 32) for more details on when and how the time step is changed. A typical increment rate is about 0.05.

Parameters for Precipitation-dependent time step control

Periods of heavy rainfall can lead to numerical instabilities if the time step is too long. To reduce the numerical instabilities, the a time step control has been introduced on the precipitation and infiltration components. You will notice the effect of these factor during the simulation by suddenly seeing very small time steps during storm events. If your model does not include the unsaturated zone, or if you are using the 2-Layer water balance method, then you can set these conditions up by a factor of 10 or more. However, if you are using the Richards equation method, then you may have to reduce these factors to achieve a stable solution.

Max precipitation depth per time step If the total amount of precipitation [mm] in the current time step exceeds this amount, the time step will be reduced by the increment rate. Then the precipitation time series will be re-sampled to see if the max precipitation depth criteria has been met. If it has not been met, the process will be repeated with progressively smaller time steps until the precipitation criteria is satisfied. Multiple sampling is important in the case where the precipitation time series is more detailed than the time step length. However, the criteria can lead to very short time steps during short term high intensity events. For example, if your model is running with maximum time steps of say 6 hours, but your precipitation time series is one hour, a high intensity one hour event could lead to time steps of a few minutes during that one hour event.

Max infiltration amount per time step If the total amount of infiltration due to ponded water [mm] in the current time step exceeds this amount, the time step will be reduced by the increment rate. Then the infiltration will be recalculated. If the infiltration criteria is still not met, the infiltration will be recalculated with progressively smaller time steps until the infiltration criteria is satisfied.

Input precipitation rate requiring its own time step If the amount of precipitation [mm] divided by the time step length [hr] in the current time step exceeds this amount, the time step will be reduced by the increment rate until this criteria is met. That is, the precipitation time series will be re-sampled with progressively smaller time steps until the precipitation rate criteria is satisfied. Multiple sampling is important in the case where the precipitation time series is more detailed than the time step length. However, the criteria can lead to very short time steps during short term high intensity events. For example, if your model is running with maximum time steps of say 6 hours, but your



precipitation time series is one hour, a high intensity one hour event could lead to time steps of a few minutes during that one hour event.

Actual time step for the different components

As outlined above the overland time step is always less than or equal to the UZ time step and the UZ time step is always less than or equal to the SZ time step. However, the exchanges are only made at a common time step boundary. This means that if one of the time steps is changed, then all of the time steps must change accordingly. To ensure that the time steps always meet, the initial ratios in the maximum time steps specified in this dialogue are maintained.

After a reduction in time step, the subsequent time step will be increased by

$$timestep = timestep \times (1 + IncrementRate) \quad (2.1)$$

until the maximum allowed time step is reached.

Relationship to Storing Time Steps

The Storing Time Step specified in the Detailed time series output (V.2 p. 138) dialogue, must also match up with maximum time steps. Thus,

- The OL storing time step must be an integer multiple of the Max UZ time step,
- The UZ storing time step must be an integer multiple of the Max UZ time step,
- The SZ storing time step must be an integer multiple of the Max SZ time step,
- The SZ Flow storing time step must be an integer multiple of the Max SZ time step, and
- The Hot start storing time step must be an integer multiple of the maximum of all the storing time steps (usually the SZ Flow storing time step)

For example, if the Maximum allowed SZ time step is 24 hrs, then the SZ Storing Time Step can only be a multiple of 24 hours (i.e. 24, 48, 72 hours, etc.)



2.2.4 OL Computational Control Parameters

OL Computational Control Parameters

Solver Type and Solver-specific Parameters:

☒ Successive Overrelaxation (SOR)

Maximum number of iterations:

Maximum head change per iteration: [m]

Maximum residual error: [m/d]

Under-relaxation factor: [0.01 - 1.0]

☐ Explicit (Recommended when overbank spilling is allowed)

Maximum courant number: [0.1 - 0.9] (for adaptive time step)

Common stability parameters:

Threshold water depth for overland flow: [m]

Threshold gradient for applying low-gradient flow reduction:

Overland-River exchange calculation:

☐ Manning equation (using OL flow Manning numbers)

☒ Weir formula: (Weir data specified in MIKE 11)

Threshold head difference for applying low-gradient flow reduction:

Overland Computational Control Parameters

Conditions: if Overland Flow specified in Model Components

Variable	Units
Maximum number of iterations	-
Maximum head change per iteration	[m]
Maximum residual error	[m/d]
Under-relaxation factor	-
Maximum courant number	-
Threshold water depth for overland flow	[m]



Variable	Units
Threshold gradient for applying low gradient flow reduction	-
Threshold head difference for applying low gradient flow reduction	[m]

Solver Type for Overland Flow

Overland flow can be solved using either using the implicit Successive Over-Relaxation (SOR) Numerical Solution (V.2 p. 216) or an Explicit Numerical Solution (V.2 p. 217)).

In the SOR method, the depth of overland flow is solved iteratively (implicitly) using a Gauss-Seidel matrix solution. The SOR method is a means of speeding up convergence in the Gauss-Seidel method. The iteration procedure is identical to that used in the saturated zone, except that no over-relaxation is allowed. The SOR method is faster but not as accurate compared to the Explicit method. However, when calculating overland flow to generate runoff, the SOR method is typically accurate enough.

The Explicit method is typically much slower than the implicit SOR method because it usually requires much smaller time steps. However, it is generally more accurate than the SOR method and is often used to calculate surface water flows during flooding. Thus, we recommend that you use the explicit method when overbank spilling from MIKE 11 to overland flow is allowed.

SOR parameters

Maximum number of iterations If the maximum number of iterations is reached, then simulation will go onto the next time step and a warning will be written to the simulation log file. The default value is 200 iterations, which is normally reasonable. You may want to increase this if you are consistently exceeding the maximum number of iterations, and the residuals are slowly decreasing.

Increasing the following two residual values will make the solution converge sooner, but the solution may not be accurate. However, it may be such that only a couple of points outside your area of interest are dominating the convergence and inaccuracies in these points may be tolerable. Decreasing these values will make the solution more accurate, but the solution may not converge at very small values and it may take a long time to reach the solution. If you decrease these values then you may also have to increase the maximum number of iterations.



Maximum head change per iteration If the difference in water level between iterations in any grid cell is greater than this amount, then a new iteration will be started. This will continue until the maximum number of iterations is reached. The default value is 0.0001 m, which is normally reasonable.

Maximum residual error If the difference in water level between iterations divided by the time step length in any grid cell is greater than this amount, then a new iteration will be started. This will continue until the maximum number of iterations is reached. The default value is 0.0001m/d, which is normally reasonable.

Under-relaxation factor The change in head for the next iteration is multiplied by the under-relaxation factor to help prevent numerical oscillations. Thus, lowering the under-relaxation factor is useful when your solution is failing to converge due to oscillations. This will have the affect of reducing the actual head change used in the next iteration. However, often it is more effective to reduce the time step. The under-relaxation factor must be between 0.01 and 1.0. The default value is 0.9, which is normally reasonable.

Explicit parameters

Maximum courant number The courant number represents the ratio of the speed of wave propagation to the grid spacing. In other words, a courant number greater than one would imply that a wave would pass through a grid cell in less than one time step. This would lead to severe numerical instabilities in an explicit solution. The courant number must be greater than 0.1 and less then 1.0. For a detailed discussion of the courant criteria see the Courant criteria (*V.2 p. 217*).

Common stability parameters

The common stability parameters are used by both the implicit SOR solver and the explicit solver.

Threshold water depth for overland flow This is the minimum depth of water on the ground surface before overland flow is calculated. Very shallow depths of water will normally lead to numerical instabilities. The default value is 0.0001 m.

The threshold depth for overland flow should not be confused with the Detention Storage (*V.2 p. 84*). The detention storage is related to the amount of water stored in local depression on the ground surface, which must be filled before water can flow laterally to an adjacent cell.



Threshold gradient for applying low gradient flow reduction In flat areas with ponded water, the head gradient between grid cells will be zero or nearly zero and numerical instabilities will be likely. To dampen these numerical instabilities in areas with low lateral gradients a damping function has been implemented. The damping function essentially increases the resistance to flow between cells. This makes the solution more stable and allows for larger time steps. However, the resulting gradients will be artificially high in the affected cells and the solution will begin to diverge from the Mannings solution. At very low gradients this is normally insignificant, but as the gradient increases the differences can become significant.

The damping function is controlled by a minimum gradient below which the damping function becomes active. Experience suggests that you can get reasonable results with a minimum gradient between 0.0005 and 0.001. The default minimum gradient is 0.001. Higher values may lead to a divergence from the Mannings solution. Lower values may lead to more accurate solutions, but at the expense of numerical instabilities, smaller time steps and longer simulation times.

For a detailed discussion of the damping function see the Low gradient damping function (*V.2 p. 218*).

Overland River Exchange Calculation

Overland flow will discharge into the MIKE 11 river link if the water elevation in the cell is higher than the bank elevation. The rate of discharge to the river is dictated by the Mannings calculation for overland flow. However, this flow is only one way, that is from overland flow to the river.

If you want to include overbank spilling from the river to the overland grid cells, then you must use the weir formula, which provides a mechanism for water to flow back and forth across the river bank.

Note Whether or not to allow overbank spilling from the river to overland flow is made in MIKE 11 for each coupling reach. If you do not allow overbank spilling in MIKE 11, then the overland river exchange is only one way, but uses the weir formula instead of the Mannings formula for calculating the amount of exchange flow.

If you do not use the overbank spilling option, then you can still use the flood inundation option to “flood” a flood plain. In this case, though, the flooding is not calculated as part of the overland flow, but remains part of the water balance of MIKE 11. For more information on the flood inundation method see the section on Area Inundation using Flood Codes (areal



source/sink) (V.2 p. 239), and the Inundation options by Flood Code (V.1 p. 173).

Threshold head difference for applying low gradient flow reduction If the difference in water level between the river and the overland flow cell is less than this threshold, then the flow over the weir is reduced to dampen numerical instabilities. In this case, the same damping function is used as in low gradient areas. The damping function essentially increases the resistance to flow between the cell and the river link. This makes the solution more stable and allows for larger time steps. However, the resulting gradients will be artificially high in the affected cells and the solution will begin to diverge from the Mannings solution. At very low gradients this is normally insignificant, but as the gradient increases the differences can become significant.

The damping function is controlled by a minimum head difference between the river and cell below which the damping function becomes active. Experience suggests that you can get reasonable results with a minimum head difference between 0.05 and 0.1 metres. The default minimum head difference is 0.1. Higher values may lead to a divergence from the Mannings solution. Lower values may lead to more accurate solutions, but at the expense of numerical instabilities, smaller time steps and longer simulation times.

For a detailed discussion of the damping function see the Low gradient damping function (V.2 p. 218).

Related Items:

- Low gradient damping function (V.2 p. 218)
- Overland Flow - Reference (V.2 p. 211)



2.2.5 UZ Computational Control Parameters

UZ Computational Control Parameters

UZ-SZ Coupling Control (Full Richards and Simple UZ)

Max. profile water balance error 0.001 [m]

Full Richards Solution

Iteration Control

Maximum no. of iterations 50

Iteration stop criteria (fraction of Psi) 0.002

Timestep Reduction Control (UZ Restart)

Max. water balance error in one node (fraction) 0.03

UZ Computational Control Parameters

Conditions: if Unsaturated Flow specified in Model Components

Variable	Units
Maximum profile water balance error	EUM [water level]
Maximum number of iterations	-
Iteration Stop criteria	-
Maximum water balance error in one node	m

The unsaturated flow is solved iteratively when the Richards Equation method is chosen, but is solved directly for both the Gravity Flow module and the Two-Layer Water Balance methods.

When the Richards Equation method is used, you can specify the iteration stop criteria and the maximum water balance error, in addition to the maximum number of iterations.

Maximum profile water balance error The tolerance criteria in the UZ-SZ coupling procedure is the maximum allowed accumulated water balance error in one UZ column. If this value is exceeded the location of the groundwater table will be adjusted and additional computations in the UZ component will be done until this criteria is met. The recom-



mended value is 0.002m or less. Since the Two-Layer Water Balance method does not calculate a water table per se, this parameter is only used in the Richards Equation method and the Gravity Flow module.

Iteration Stop Criteria The solution is deemed to have converged when the difference in pressure head between iterations for all nodes is less than or equal to the iteration stop criteria. The recommended value is 0.02m or less.

Maximum water balance error in one node This is defined as the fraction of the total saturated volume in the node. the time step will automatically be reduced if the error is exceeded. The recommended value is 0.01 or less, with a value of 0.001 to 0.002 m being reasonable.

2.2.6 SZ Computational Control Parameters

SZ Computational Control Parameters

Solver Type

- ☒ Preconditioned Conjugate Gradient, Transient
- ☐ Preconditioned Conjugate Gradient, Steady State
- ☐ Successive Overrelaxation Package (SOR)

SZ Computational Control Parameters

Conditions: if Saturated Flow specified in Model Components

For the saturated zone in MIKE SHE there are two solvers to choose from:

- the pre-conditioned conjugate gradient method, and
- the successive over-relaxation method.

The Successive Over-relaxation solver is the original solver in MIKE SHE and the Pre-conditioned Conjugate Gradient Solver is based on the USGS's PCG2 solver for MODFLOW (Hill, 1990).

Steady-state vs Transient Simulations

The Solver type controls whether or not the simulation is run as a Steady-state model or not - if you chose the Pre-conditioned Conjugate Gradient-Steady-State option then the simulation will be run in steady-state. Otherwise, the simulation will be run as a transient simulation.



If the SZ simulation is steady-state, then the PCG solver is the only solver available. Although the same options are available for both the steady-state and the transient PCG solvers the optimal parameters or combination of parameters and options is most likely different in the two cases. Thus, the recommended settings are different in both cases.

Iteration Control

Variable	Units
Maximum number of iterations	-
Maximum head changed per iteration	EUM [elevation]
Maximum residual error	[m/day]

The iteration procedure can be stopped when either the iteration stop criteria are reached or when the maximum number of iterations is reached. The iteration stop criteria consist of a mass balance criteria and a head criteria. Both of these criteria must be chosen carefully to ensure that the solution has converged to the correct solution.

The default option settings normally perform well in most applications. Usually there is no need for changes. Changes to the default options should not be done unless the solution does not converge or convergence is extremely slow.

Maximum number of iterations - The maximum number of iterations should be sufficiently large to avoid water balance errors due to non-convergence.

Maximum head change per iteration - The head criteria determines the accuracy of the solution. The computational time is very dependent on the value used. A value of 0.01m (0.025ft) is usually sufficient. During the initial model calibration a higher stop criteria can be used. The sensitivity of the head stop criteria should always be examined.



Maximum residual error - The maximum residual error is the tolerable mass balance error, which should be low but sufficiently high that the number of iterations is not excessive. A value of 0.001m/d is usually good for regional groundwater studies. In smaller scale applications, where solute transport will be investigated the mass balance criteria should be reduced, for example, to 0.0001 or 0.00001m/d. In general, a larger mass balance criteria should be used during model calibration to keep the initial simulation times shorter. For scenario calculations, the mass balance criteria can be reduced to ensure more accurate simulations and smaller mass balance errors. The SZ water balance should always be checked at the end of the simulation to ensure that the mass balance criteria used was reasonable.

Sink de-activation in drying cells

Sink de-activation in drying cells

Saturated thickness threshold	<input type="text" value="0.05"/>	[m]
-------------------------------	-----------------------------------	-----

Variable	Units
Saturated thickness threshold	EUM [water level]

Saturated thickness threshold - To avoid numerical stability problems the minimum depth of water in a cell should always be greater than zero. However, if the water depth is close to zero, then sinks in the cell, such as wells, should be turned off, since the cell is effectively 'dry'. This value is the minimum depth of water in the cell and the depth at which the sinks are deactivated.

Maximum exchange from river during one time step

Maximum exchange from river during one time step

Max. fraction of H-point volume	<input type="text" value="0.9"/>
---------------------------------	----------------------------------

Variable	Units
Maximum fraction of H point volume	-



Max. fraction of H-point volume - If you are simulating rivers with MIKE 11, then this represents the maximum water that can be removed from the river during one SZ time step. Removing larger amounts of water could effectively dry out the river. If this occurs, then the SZ solver will issue a warning and only this fraction of water will be removed, which prevents rapid drying out of the river during a single time step.

Pre-conditioned Conjugate Gradient

Advanced Settings

Gradual activation of SZ drainage - To prevent numerical oscillations the drainage constant may be adjusted between 0 and the actual drainage time constant defined in the input for SZ drainage. The option has been found to have a dampening effect when the groundwater table fluctuates around the drainage level between iterations (and does not entail reductions in the drain flow in the final solution). For the steady-state solver and the transient solver the option is by default turned ON.

Horizontal conductance averaging between iterations - To prevent potential oscillations of the numerical scheme when rapid changes between dry and wet conditions occur a mean conductance is applied by taking the conductance of the previous (outer) iteration into account. By default this option is enabled for both steady-state and transient simulations.

Under-relaxation - Under-relaxation factors can be calculated automatically as part of the outer iteration loop. The algorithm determines the factors based on the minimum residual-2-norm value found for 4 different factors. To avoid numerical oscillations the factor is determined as 90% of the factor used in the previous iteration and 10% of the current optimal factor.

The second option is to define a constant relaxation factor between 0 and 1. In general a low value will provide convergence, but at a low



convergence rate - i.e. with many SZ iterations. Higher values increases the convergence rates, but also the risk of non-convergence. As a general rule a value of 0.2 has been found suitable for most set-ups.

The time used for automatic estimation of relaxation factors may be significant compared to subsequently solving the equations and the option is only recommended in steady-state cases. In transient simulations, 'No under-relaxation' is recommended.

Successive Over-relaxation

Over-relaxation

Relaxation factor (1.0-2.0):

Variable	Dimensions
Relaxation Factor	-

Over-relaxation

Relaxation factor - The speed of convergence also depends on the relaxation coefficient. Before you set up your model for a long simulation, you should test the iteration procedure by running a few short simulations with different relaxation coefficients. This coefficient must be between 1.0 and 2.0, with a typical value between 1.3 and 1.6.



2.3 Water Quality Simulation Specification

WQ Simulation Specification		
	WM	AD
Overland Flow (OL):	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>
River and Lakes	<input type="checkbox"/>	<input type="checkbox"/>
Unsaturated Flow (UZ):	<input type="checkbox"/>	<input type="checkbox"/>
Evapotranspiration (ET):	<input type="checkbox"/>	<input type="checkbox"/> Plant uptake
Saturated Flow (SZ):	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>

WQ Simulation Specification

Conditions: if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue

In the WQ Simulation Specification dialogue, you can select which components of the hydrologic cycle will be included in the water quality simulation. The advection dispersion method calculates solute movements based on the intercell flows calculated in a water movement simulation. Therefore, only those components that are included in the water movement solution can be selected.

Solute transport in surface water bodies is specified in and calculated by MIKE 11.

If selected, plant uptake by roots is treated as a solute sink in the unsaturated zone.

Note In the initial release of the 2007 Release, the advection dispersion module is only available for the saturated zone (SZ) and the overland flow (OL) components. Solute transport in the unsaturated zone, as well as solute sorption and decay will be added to the MIKE SHE user interface in a service pack in 2007. In the mean time, if you need to simulate solute movement in the unsaturated zone or sorption and decay, then you can specify the input files manually and use the command line interface to run



the executable. For a detailed description on how to do this, refer to Using the Fully Integrated AD Module (*V.1 p. 197*). Also available from the command line is the random walk particle tracking method, which is described in Working with Particle Tracking (*V.1 p. 229*).

2.3.1 *WQ Simulation Title*

WQ Simulation Title

Simulation Title:

Simulation Description:

WQ Simulation Title	
Conditions:	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue

Title and Description - The Title and Description will be written to output files and appear on plots of the simulation results.



2.3.2 WQ Simulation Period

WQ Simulation Period

WQ Simulation Period

Start Date: 2000/01/01 00:00

End Date: 2000/02/01 00:00

Flow Results for Water Quality Simulation

☐ No recycling on flow results

☒ Recycling on flow results

Cycle Restart Date: 2000/01/01 00:00

Cycle End Date: 2000/02/01 00:00

☐ Constant Water Movement Flow Field

Date for Flow Field Solution: 2000/02/01 00:00

WQ Simulation Period

Conditions: if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue

WQ Simulation Period

The water quality simulation does not have to be the same length as the water movement simulation. The only restriction is that the start date for the water quality simulation must be within the water movement simulation.

Flow Results for Water Quality Simulation

A water quality simulation requires the cell-by-cell water fluxes calculated by the water movement simulation. However, the water quality simulation does not have to be the same period as the water movement simulation. Therefore, the user interface is flexible in how it will use water movement cell-by-cell flow data.



No recycling on results - In this case, the water quality simulation end date must also be within the water movement simulation period, which means that the water quality simulation cannot extend beyond the water movement simulation.

Recycling on flow results - In this case, the water quality simulation can be much longer than the water movement simulation, based on a repeated set of water movement results. The water quality simulation starts on the Start Date with the flow results from the Cycle Restart Date. When the water quality simulation period reaches the Cycle End Date, the WQ simulation will continue but the flow results will be restarted at the Cycle Restart Date.

If the recycle dates do not match one of the saved time steps, then the nearest saved time step is used.

For example, you may have a two-year water movement simulation but you may want to simulate water quality for 10 years. To do this, you would specify the start and stop dates of the part of the water movement simulation that you want repeated. If you want to repeat the whole water movement simulation, then you would specify the beginning and end of the water movement simulation.

Constant water movement flow field - In this case, the nearest saved time step to this date will be used as a steady-state flow field for the transient water quality simulation.



2.3.3 Water Quality Time Step Control

WQ Time Step Control			
	Saturated Zone (SZ)	Unsaturated Zone (UZ)	Overland (OL)
Max. Simulation timestep:	<input type="text" value="1000000000"/>	<input type="text" value="1000000000"/>	<input type="text" value="1000000000"/> [hrs]
Max. Advective Courant Number:	<input type="text" value="0.8"/>	<input type="text" value="0.8"/>	<input type="text" value="0.8"/> [-]
Max. Dispersive Courant Number:	<input type="text" value="0.5"/>	<input type="text" value="0.5"/>	<input type="text" value="0.5"/> [-]
Max. Transport Limit:	<input type="text" value="0.95"/>	<input type="text" value="0.95"/>	<input type="text" value="0.95"/> [-]

Water Quality Time Step Control

Conditions: if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue

The water quality simulation is completely decoupled from the water movement simulation and like the water movement itself, the water quality time steps can be different in each of the overland flow, unsaturated flow and saturated flow.

Maximum Simulation Time Step - This is the maximum user-specified time step allowed. The default value is very high so that the simulation runs by default with the highest possible time step. You might want to set this value to a short time interval, if you want the WQ time step to be uniform during the WQ simulation.

Stability Criteria

The courant number is a measure of the ratio of flow rate to grid size. For numerical stability, it is important that the solutes do not travel too far in one time step. The time step is reduced until all the time step criteria below are met.

Max. Advective Courant Number - The advective courant number represents the ratio of how fast a particle moves in the flow field to the cell size. This criteria is likely to be controlling if your flow velocities are high, or your dispersivity values are very low or zero. The default value is 0.8. If your actual time step is being controlled by this criteria, then you could increase it to make the simulation run faster. However,

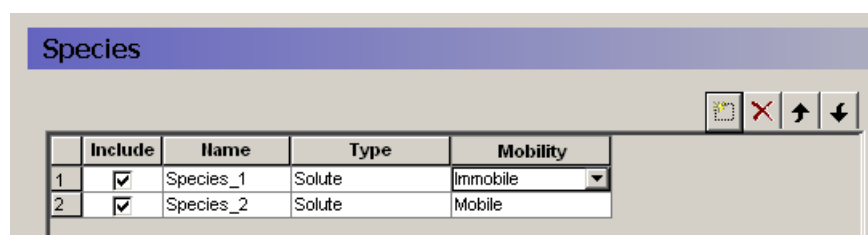


you will need to check to make sure the simulation has converge properly and that the mass balance is reasonable.

Max. Dispersive Courant Number - The dispersive courant number represents the ratio of how fast a particle moves across a cell due to dispersion to the cell size. This criteria is likely to be controlling when the velocities are very slow and the dispersivity is non-zero. The default value is 0.5. If your actual time step is being controlled by this criteria, then you could increase it to make the simulation run faster. However, you will need to check to make sure the simulation has converge properly and that the mass balance is reasonable.

Max. Transport Limit - The transport limit restricts the fraction of the total amount of mass that can leave the cell in one time step. The default value 0.95, which usually does not limit the time step.

2.4 Species



	Include	Name	Type	Mobility
1	<input checked="" type="checkbox"/>	Species_1	Solute	Immobile
2	<input checked="" type="checkbox"/>	Species_2	Solute	Mobile

Species

Conditions: if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue

In this dialogue, you add species by clicking on the Insert icon. You delete species by selecting the species from the table and clicking on the Delete icon.

The table includes the list of species for the WQ simulation and the physical properties of the chemical species.

Include - Turning off the include checkbox allows you to exclude a species from the simulation without having to remove it and all of its accompanying sources, etc.



Name - This is the displayed identifier in all subsequent dialogues and in the data tree.

Type - At the moment, Solute is the only active type. However, in future versions, other types may be available.

Mobility - The mobility refers to the ability of the species to move with the water. A species can be Mobile or Immobile. The distinction is made because mobile and immobile versions of the same chemical can have different reaction rates, such as half life. Also, when a mobile species absorbs to the soil grains, then it becomes part of the immobile version of that species.

2.5 Species Parameters

Species Params						
	Name	Reference Temperature	Temp-decay exponent	Water content decay	Plant uptake transpiration factor	Solubility
1	NaCl	20	0.1	0	0.1	3.4

Species Parameters

Conditions: if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue

Reference Temperature - The reference temperature is used for the temperature dependent decay rate calculations.

Temp-decay exponent - This is the exponent used in the temperature dependent decay rate calculations.

Water content decay exponent - This is the exponent used in the water content decay rate calculations for the unsaturated zone.

Plant uptake transpiration factor - This is the factor that determines the rate at which plants will remove the mobile solute from the water.

Solubility - Since evaporation can cause the overland concentration to increase, solubility needs to be specified to avoid unrealistic high con-



centrations. The species precipitates if the concentration exceeds the solubility. The precipitate dissolves again if the concentration falls below the solubility. The solubility is a uniform value per species with units of $[g/m^2]$

2.6 Model Domain and Grid

Model Domain and Grid

☒ Catchment defined by Dfs File ☐ Catchment defined by Shape File

C:\5.Testing\NRSøby\MAPS\innocat3c.dfs ... Edit... Create...

Catchment size and origin

NX	NY	Cell Size	X0	Y0
330	400	250 [m]	543000 [m]	6065500 [m]

Model Domain and Grid

☐ Catchment defined by Dfs File ☒ Catchment defined by Shape File

C:\5.Testing\NRSøby\GIS\model_area.shp ... Edit... Create...

Shape File Unit of X- and Y-axes: meter

Catchment size and origin

NX	NY	Cell Size	X0	Y0
90	90	250 [m]	578000 [m]	6118000 [m]

Model Domain and Grid

dialogue Type: Special; Integer Grid Codes

EUM Data Integer Grid Code
Units

Valid Values -1e-35, 1, 2

Regardless of the components included in your model, the first step in your model development is to define the model area. On a catchment scale, the model boundary is typically a topographic divide, a groundwater divide or some combination of the two. In general, there are no constraints on the definition of the model boundaries. However, the model boundaries should be chosen carefully, keeping in mind the boundary conditions that will be used for both the surface water and groundwater components.

Any non-gridded data, such as .shp file and xyz ASCII files are interpolated to the grid defined in this dialogue.



The grid defined in this dialogue is the primary grid. Other .dfs2 gridded data does not have to use the same grid. However, if another .dfs2 file uses a different grid then Real data is interpolated. If the two grids are coincident, that is the cells are the same size and the grids line up, then the data is bilinearly interpolated to the Model Grid. If the grids are not coincident, then the data is treated as if it were point values (i.e. the same as XYZ or .shp data). Integer Grid Code must use coincident grids, as it is impossible to interpolated integer values.

Using a dfs2 file

If you define your model domain using a dfs2 grid file, then you must define the cell values as follows:

- Grid cells outside of the model domain must be assigned a delete value - usually -1e-35.
- Grid cells inside the model domain must be assigned a value of 1.
- Grid cells on the model boundary must be assigned a value of 2.

This distinction between interior grid cells and boundary cells is to facilitate the definition of boundary conditions. For example, drainage flow can be routed to external boundaries but not to internal boundaries.

The catchment definition is displayed in the greyed out text boxes but is not editable, since the catchment definition is part of the .dfs2 file format. If you want to change the cell size, origin, number of cells etc., you must change the .dfs2 file itself. For more information on editing and setting up the Model Domain and Grid see Your Conceptual Geologic Model (V.1 p. 49).

Using a shp file

It is much easier to define your Model Domain and Grid via an ArcView .shp file (i.e a grid independent polygon). In this case, the definition of integer code values is taken care of automatically. Further, the definition of the grid (number of rows and columns, cell size and origin) can be easily adjusted.



2.7 Subcatchments

Subcatchment

Name: Speed River basin Grid code value: 7

☒ Use default river links

Subcatchments

Conditions:	when either of the subcatchment-based methods for Overland Flow or Saturated Flow are selected in the Simulation Specification dialogue
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The Subcatchment item appears whenever you select one of the sub-catchment based methods - Simplified Overland Flow Routing or the Linear Reservoir Method for groundwater flow.

The subcatchment items are used to identify the hydrologic subwatersheds in your model domain. For the Simplified Overland Flow Routing, the calculated overland flow in the Overland Flow Zones flows from one zone to the next within the Subcatchment. For the Linear Reservoir Method for groundwater flow, the calculated interflow is routed from one zone to the next within the Subcatchment.

For each unique integer code in the main Subcatchment map view, an additional data item is added to the data tree. In each of these sub-items, there is only one additional variable - a checkbox for using the default river links.

Use default river links - in most cases you will link the simplified overland flow and the groundwater interflow to all of the river links found in the lowest topographic zone, or the lowest interflow zone in the subcatchment. However, in some cases you may want to link the flow to particular river links. For example, if your MIKE 11 river network does not extend into the subcatchment, you can specify that the interflow discharges to a particular node or set of nodes in a nearby river network.

If you uncheck this checkbox, a sub-item will appear where you can specify the river branch and chainage to link the subcatchment to.



The river links for the baseflow zones are specified separately in the baseflow zone dialogues.

Related Items

- Simplified Overland Flow Routing (*V.2 p. 220*)
- Overland Flow Zones (*V.2 p. 87*)
- Linear Reservoir Method (*V.2 p. 307*)
- Interflow Reservoirs (*V.2 p. 105*)

2.7.1 River Links

	Branch name	Upstream	Downstream
1	Speed River	12300	14500

If you have unselected the Use default river links option in the Subcatchments dialogue or in the Baseflow Reservoirs dialogue, then this dialogue will be added to the data tree. In this dialogue, you can specify the branch name to connect a subcatchment to, as well as the upstream and downstream chainage of the branch.

This dialogue is not intelligent, in the sense that it does not read the MIKE 11 river network file. You must type the branch name exactly as it appears in the river network file and specify valid chainages. If either the name or the chainages are invalid, then you will get an error during the model pre-processing stage.

Related Items

- Coupling MIKE 11 and MIKE SHE (*V.1 p. 165*)



2.8 Topography

Topography

dialogue Type: Stationary Real Data

EUM Data Elevation
Units

In MIKE SHE, the topography defines the upper boundary of the model. The topography is used as the top elevation of both the UZ model and the SZ model. The topography defines the drainage surface for overland flow.

Many of the elevation parameters can be defined relative to the topography, such as

- Lower Level (Geological Layer or Lense, or Water Quality Layer) (*V.2 p. 111*),
- Upper Level (*V.2 p. 111*),
- Lower Level (Numerical Layer) (*V.2 p. 117*),
- Initial Potential Head (*V.2 p. 117*), and
- Drain Level (*V.2 p. 125*).

Depth parameters, such as ET Surface Depth (*V.2 p. 100*), are also measured from the topography.

Topography is typically defined from a DEM, defined from either a point theme shape file, or an ASCII file. If you have an ArcGIS Grid DEM, this can be converted to a dfs2 file using the MIKE Zero Toolbox. Surfer Grid files can be saved as an ASCII xyz files and then interpolated in MIKE SHE.

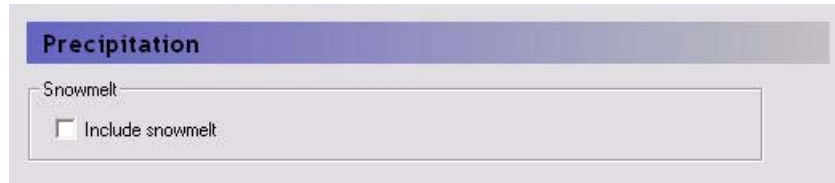
Non-dfs2 files or dfs2 files that have a different grid definition than the model grid are all interpolated to the grid defined in the Model Domain and Grid.

The bilinear interpolation method is useful for interpolating previously gridded DEM data. Whereas, the triangularisation method is useful for contour data digitized from a DEM.

**Related Items**

- Bilinear Interpolation (*V.1 p. 260*)
- Triangular Interpolation (*V.1 p. 264*)

2.9 *Precipitation*



Precipitation is always included in the data tree, but, there is only one option. That is to include or not include Snowmelt.

Snowmelt

If Snowmelt is included, then Evapotranspiration must also be included.

If Snowmelt is included then two additional dialogues are added to the data tree - one for the Snowmelt Constants and the other for the distributed Air Temperature.

In MIKE SHE Snowmelt is calculated on the basis of the degree-day using a simple method that only requires the air temperature, a degree-day factor [mm snow/day/degree C] and a threshold temperature defined the temperature at which melting occurs (usually 0 C)

Related Items

- Time Series Data (*V.1 p. 243*)
- Snowmelt Constants (*V.2 p. 62*)
- Air Temperature (*V.2 p. 60*)



2.9.1 Precipitation Rate

Precipitation Rate	
dialogue Type:	Time-varying Real Data
EUM Data Units	Grid Code
Time Series	Precipitation Rate [e.g. mm/hr], or
EUM Data Units	Rainfall [e.g. mm]

The precipitation rate is the measured rainfall and snowfall.

You can specify the precipitation rate as a rate, for example in [mm/hr], or as an amount, for example in [mm]. If you use the amount method, MIKE SHE will automatically convert this to a rate during the simulation.

If you use a rate, then the EUM Data Units must be Precipitation and the time series must be Mean Step Accumulated (*V.1 p. 247*).

If you use an amount, then the EUM Data Units must be Rainfall and the time series must be Step Accumulated (*V.1 p. 247*).

The Precipitation Rate item comprises both a distribution and a value. The distribution can be either uniform, station-based or fully distributed. If the data is station-based then for each station a sub-item will appear where you can enter the time series of values for the station.

If Snowmelt is included and the Air Temperature is below the Threshold melting temperature then the precipitation will accumulate as snow.

Related Items

- Creating Time Series in MIKE SHE (*V.1 p. 243*)
- Working with Spatial Time Series (*V.1 p. 245*)
- Time Series Types (*V.1 p. 246*)



2.9.2 Net Rainfall Fraction

Net Rainfall Fraction	
Conditions	If Evapotranspiration is NOT selected
dialogue Type	Stationary Real Data
EUM Data	Fraction
Units	

The Net Rainfall Fraction is the fraction of rainfall that is available for infiltration and overland flow. It is used to account for leaf interception and evapotranspiration when ET is not explicitly simulated.

The net recharge to the groundwater table, R_{net} , is

$$R_{net} = Prec \cdot Rainfall_{net} \cdot Infil_{frac} \quad (2.2)$$

where $Prec$ is the actual precipitation, $Rainfall_{net}$ is the Net Rainfall Fraction, and $Infil_{frac}$ is the Infiltration fraction.

Related Items

- Precipitation Rate (V.2 p. 58)
- Infiltration Fraction (V.2 p. 59)

2.9.3 Infiltration Fraction

Infiltration Fraction	
Conditions	If Overland flow is simulated but unsaturated flow is NOT simulated.
dialogue Type	Stationary Real Data
EUM Data	Fraction
Units	

The Infiltration Fraction is the fraction of ponded water that infiltrates. It is used when the unsaturated zone is not explicitly simulated.

Normally the unsaturated zone simulation calculates the amount of infiltration from overland flow, since the amount of infiltration depends on the



water content of the upper most soil horizon. If the soil is saturated, then the infiltration will be low. If the soil is very dry, then the infiltration could be very high.

However, the Net Infiltration Fraction is a stationary variable. The only way to simulate the dynamic changes in the amount of infiltration is to simulate the unsaturated zone.

Note When MIKE 11 is used in MIKE SHE, overland flow must always be included. If you want to simulate strictly saturated flow coupled to MIKE 11, then you will need to use the Infiltration Fraction instead of the unsaturated flow.

The net recharge to the groundwater table, R_{net} , is

$$R_{net} = Prec \cdot Rainfall_{net} \cdot Infil_{frac} \quad (2.3)$$

where $Prec$ is the actual precipitation, $Rainfall_{net}$ is the Net Rainfall Fraction, and $Infil_{frac}$ is the Infiltration fraction.

Related Items

- Precipitation Rate (V.2 p. 58)
- Infiltration Fraction (V.2 p. 59)

2.9.4 Air Temperature

Air Temperature	
Conditions	if Snowmelt selected
dialogue Type	Time-varying Real Data
EUM Data Units	Grid Code
Time Series EUM Data	Temperature; Instantaneous Units

This is the temperature in Celsius that is used to calculate the amount of snow that melts per time step.



Snowmelt

If the air temperature is above the Threshold melting temperature (see Snowmelt Constants) then the snow will begin to melt. The snow storage will be reduced by

$$q_{snow} = Degree_day_factor * (Air_temp - Threshold_temp) * \Delta t \quad (2.4)$$

where the *Degree day factor* and the *Threshold Temperature* are defined in the Snowmelt Constants dialogue and Δt the length of the UZ/ET time step. Snow storage will be reduced to zero if q_{snow} is greater than the snow storage.

If the air temperature is below the Threshold melting temperature, then the ET module will remove water from the snow storage as sublimation before any other ET is removed using

$$E_{snow} = Reference_ET * \Delta t \quad (2.5)$$

where Reference_ET refers to the Reference Evapotranspiration before being reduce by the Crop Coefficient, k_c , that is specified in the Vegetation Development Table (V.2 p. 180). If there is not enough snow storage then E_{snow} will reduce the snow storage to zero.

Air temperature must be instantaneous values. Thus, for air temperature time series, an average air temperature is used in each time step based on a linear interpolation between two instantaneous values.

Related Items

- Snowmelt Constants (V.2 p. 62)
- Creating Time Series in MIKE SHE (V.1 p. 243)
- Working with Spatial Time Series (V.1 p. 245)
- Time Series Types (V.1 p. 246)



2.9.5 Snowmelt Constants

Snowmelt Constants

Snowmelt Constants

Degree-day factor mm/day/°C

Threshold melting temperature °C

Degree-day factor

The degree day factor [mm snow/day/degree C] is the amount of snow that melts per day for every degree the Air Temperature (*V.2 p. 60*) is above the threshold melting temperature.

Threshold melting temperature

The threshold melting temperature is the temperature at which the snow starts to melt - usually 0C.

Distributed snowmelt constants can be specified using the Extra Parameters section. See Distributed Snow Melt Constants (*V.1 p. 157*).

Related Items

- Distributed Snow Melt Constants (*V.1 p. 157*)
- Air Temperature (*V.2 p. 60*)

2.10 Land Use

Land Use

☒ Paved areas
☐ Check water level before routing to river

☒ Irrigation (Requires ET and UZ)

Priority scheme:

The Land Use item in the data tree is used to define the items that are on the land surface that affect the hydrology in your model area, including

- Vegetation distribution,
- Paved areas (Paved Runoff Coefficient (*V.2 p. 68*)), and
- Irrigation.



Paved areas - The Paved areas option allows you to direct a portion of the overland flow directly to the SZ drainage network. Activating the Paved areas option, creates a sub-tree with the Paved Runoff Coefficient (V.2 p. 68). The Paved areas option is available only when Overland flow is simulated.

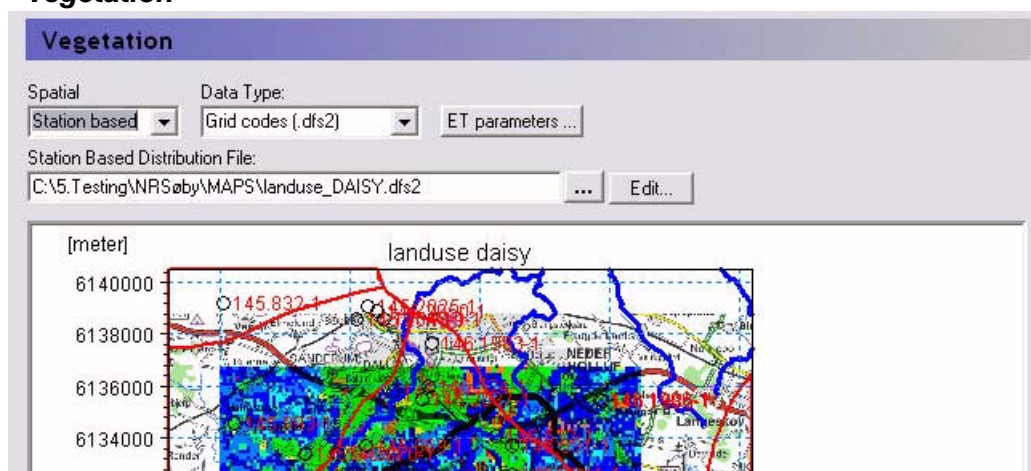
Check water level before routing to river - If this option is checked, then MIKE SHE checks to make sure that the water level in the receiving river is lower than the drain level in the current cell. If the river is higher than the drain level then no paved runoff will occur.

Irrigation - The Irrigation option allows you to specify a demand driven irrigation scheme with priorities. Activating the Irrigation option creates several sub-items in the data tree for the irrigation parameters. The Irrigation option requires that both Evapotranspiration and Unsaturated Flow be simulated. For more information see Irrigation Command Areas (V.2 p. 69)

Priority Scheme - The priority scheme is used by the Irrigation module to rank the model areas in terms of priority for irrigation. Two options are allowed: Equal Volume or Equal Shortage. If the water is to be distributed based on equal volume, then all cells with the same priority number will receive an equal amount of water, regardless of their actual demand. If the water is to be distributed based on equal shortage, then all cells with the same priority number will receive an amount of water that satisfies an equal percentage of their actual demand. For more information see Irrigation Priorities (V.2 p. 78)



2.10.1 Vegetation



Vegetation

Conditions:	If Evapotranspiration selected in the Simulation Specification dialogue
EUM Data Units	Grid Code
Time Series EUM Data Units	Leaf Area Index and Root Depth, or Vegetation Property File (see ET Vegetation Properties Editor (V.2 p. 177))
dialogue Type:	Special version of Time-varying Real Data

The main Vegetation dialogue is used to define the distribution of vegetation across your model area. It works the same as any other dialogue for Grid Codes with associated time series data. In this case, however, there are two relevant time series parameters: the Leaf Area Index and the Root Depth. Both of these parameters can be defined as constants, via .dfs0 files, or they can be defined from a Vegetation Properties file.

Note. The crop coefficient, K_c , is only available in the Vegetation Properties file.

Using a Vegetation Properties file

The Vegetation Properties file typically contains a time series of the root depth and leaf area index for either one year or for the growing season. If you are using a properties file, then you have to specify the crop development schedule.

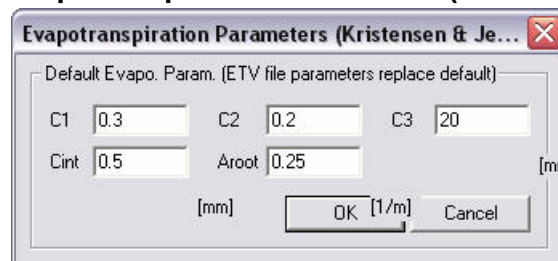


Name - This is the name of the crop or vegetation type in the properties file. This name must match exactly the name in the properties file. When you select the file name using the browse button, [...], you can select the vegetation item from a list of available vegetation types found in the file.

Start Date - The vegetation properties file typically contains information on a year or growing season basis - starting from Day 1. Thus, the Start Date is the calendar date for the beginning of the growing year or growing season. If you want to simulate consecutive growing seasons, you must re-enter the start dates appropriately. If data is missing in the time series, then the Leaf Area Index and the Root Depth are both assumed to equal zero. If the start dates overlap with the growing season information in the vegetation database, a warning will be issued in the log file that says the crop development was not over yet before the new crop was started. MIKE SHE will then start a new crop cycle at the new start date.

The Vegetation dialogue also includes a button for the Evapotranspiration Parameters (Kristensen and Jensen), which are global stationary parameters.

Evapotranspiration Parameters (Kristensen and Jensen)



The Vegetation dialogue also includes a button labelled **ET Parameters...** Clicking on this button pops up the dialogue for specifying the default ET parameters for the Kristensen and Jensen model. The Evapotranspiration parameters in this dialogue do not vary in time and are global for the model. However, if the Vegetation Properties file option is used for the Leaf Area Index and Root Depth parameters, these values can be overridden by crop specific values specified in the vegetation properties file.

The following sections are condensed from the description of the Kristensen and Jensen method in the Evapotranspiration - Reference chapter, which should be consulted for more detailed information.

Interception Coefficient, C_{int} - The interception process is modelled as an interception storage, which must be filled before stem flow to the



ground surface takes place. The coefficient C_{int} defines the interception storage capacity of the vegetation per unit of LAI. A typical value is about 0.05 mm but a more exact value may be determined through calibration (see Equation (15.2)).

Note The interception storage is calculated each time step and the rate of evaporation is usually high enough to remove all the interception storage in each time step. Thus, the total amount of water removed from interception storage depends on the length of the time step. This can lead to confusion when comparing water balances, if the time steps are different.

C₁, C₂ and C₃

The equations for actual transpiration, Eq. (15.3), and soil evaporation, Eq. (15.8), contain three empirical coefficients, C_1 , C_2 , and C_3 . The coefficients C_1 and C_2 are used in the transpiration function, $f_1(\text{LAI})$ (Eq.(15.4)). C_3 is also part of Eq. (15.3), but is the only variable found in the soil moisture function, Eq. (15.5).

C₁ - C_I is plant dependent. For agricultural crops and grass, C_I has been estimated to be about 0.3. C_I influences the ratio soil evaporation to transpiration. This is illustrated in Figure 15.7. For smaller C_I values the soil evaporation becomes larger relative to transpiration. For higher C_I values, the ratio approaches the basic ratio determined by C_2 and the input value of LAI.

C₂ - For agricultural crops and grass, grown on clayey-loamy soils, C_2 has been estimated to be about 0.2. Similar to C_1 , C_2 influences the distribution between soil evaporation and transpiration, as shown in Figure 15.8. For higher values of C_2 , a larger percentage of the actual ET will be soil evaporation. Since soil evaporation only occurs from the upper most node (closest to the ground surface) in the UZ soil profile, water extraction from the top node is weighted higher. This is illustrated in Figure 15.8, where 23 per cent and 61 per cent of the total extraction takes place in the top node for C_2 values of 0 and 0.5 respectively.

Thus, changing C_2 will influence the ratio of soil evaporation to transpiration, which in turn will influence the total actual evapotranspiration possible under dry conditions. Higher values of C_2 will lead to smaller values of total actual evapotranspiration because more water will be extracted from the top node, which subsequently dries out faster. Therefore, the total actual evapotranspiration will become sensitive to the ability of the soil to draw water upwards via capillary action.



C_3 - C_3 has not been evaluated experimentally. Typically, a value for C_3 of 20 mm/day is used, which is somewhat higher than the value of 10 mm/day proposed by Kristensen and Jensen (1975). C_3 may depend on soil type and root density. The more water released at low matrix potential and the greater the root density, the higher should the value of C_3 be. Further discussion is given in Kristensen and Jensen (1975).

Root Mass Distribution Parameter, *AROOT*

Water extraction by the roots for transpiration varies over the growing season. In nature, the exact root development is a complex process, which depends on the climatic conditions and the moisture conditions in the soil. Thus, MIKE SHE allows for a root distribution determined by the root depth (time varying) and a general, vertical root-density distribution, defined by *AROOT*, see Figure 15.3. In the above dialogue, *AROOT* is not time varying, but can be specified as a time series using the ET Vegetation Properties Editor (V.2 p. 177).

How the water extraction is distributed with depth depends on the *AROOT* parameter. Figure 15.4 shows the distribution of transpiration for different values of *AROOT*, assuming that the transpiration is at the reference rate with no interception loss ($C_{int}=0$) and no soil evaporation loss ($C_2=0$). The figure shows that the root distribution, and the subsequent transpiration, becomes more uniformly distributed as *AROOT* approaches 0. During simulations, the total actual transpiration tends to become smaller for higher values of *AROOT* because most of the water is drawn from the upper layer, which subsequently dries out faster. The actual transpiration, therefore, becomes more dependent on the ability of the soil to conduct water upwards (capillary rise) to the layers with high root density.

Figure 15.5 shows the effect of the root depth, given the same value of *AROOT*. A shallower root depth will lead to more transpiration from the upper unsaturated zone layers because a larger proportion of the roots will be located in the upper part of the profile. However, again, this may lead to smaller actual transpiration, if the ability of the soil to conduct water upwards is limited.

Thus, *AROOT* is an important parameter for estimating how much water can be drawn from the soil profile under dry conditions.

Related Items

- Kristensen and Jensen method (V.2 p. 244)
- ET Vegetation Properties Editor (V.2 p. 177)



2.10.2 Paved Runoff Coefficient

Paved Runoff Coefficient	
Conditions:	If Overland Flow selected in the Simulation Specification dialogue and Paved Areas selected in Land Use dialogue
dialogue Type:	Stationary Real Data
EUM Data Units	Fraction

The Paved Runoff Coefficient defines the fraction of overland flow that drains to storm sewers and other surface drainage features in paved areas. The Paved Runoff Coefficient acts in two ways:

- 1 it tells MIKE SHE where there is paving, and
- 2 the value specifies how much of the overland flow is allowed to infiltrate and how much should be 'drained away'.

Thus, in non-paved cells, you should use a value of zero or an Undefined Value (-1e-35). Whereas, in paved cells, you must specify a value between 0 and 1. The coefficient value defines the fraction of the overland flow that will be drained via the SZ drainage network in the current time step. However, the water is not added to the water in the drains, but rather it is sent directly to the boundary specified via the SZ drainage network.

For example, most rainfall on a paved surface is drained to storm sewers and the storm sewers typically drain directly in a river or stream. Thus, if 25% of your land area is paved, then a paved runoff coefficient of 0.25 will remove 25% of the overland flow and send it directly to the river link specified for the local SZ drainage network. The remaining 75% will be available for infiltration in the current time step and what does not infiltrate will flow as overland flow to the adjacent cells.

Check water level before routing to river - If this option is checked in the Land Use dialogue, then MIKE SHE checks to make sure that the water level in the receiving river is lower than the drain level in the current cell. If the river is higher than the drain level then no paved runoff will occur.



2.10.3 Irrigation Command Areas

Irrigation command area

ID:

Sources:

	Type	Water Application
1	River	Sprinkler
2	Single well	Drip
3	Shallow well	Sheet
4	External	Sprinkler

Shallow well source:

Irrigation Command Areas

Conditions:	Irrigation selected in Land Use and UZ and ET simulated
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The Irrigation Command Areas are used to describe where the water comes from and how the irrigation water is applied to the model.

The Irrigation Command Area data item is divided into two dialogues. The first is the distribution dialogue for the Command Areas and the second is the Water source and Application method for each of the command areas.

Each source can also be limited by a licensed maximum amount of water in any period - License Limited Irrigation (V.2 p. 75).

Calculation sequence and shortage handling for SZ Linear Reservoirs

For each rank (no. of ranks = max no. of sources specified for any command area) and each Priority (usually only 1) do the following:

- 1 Calculate the total demand of remote- and shallow well sources of the actual Rank and Priority from all baseflow reservoirs (1 & 2).
- 2 Calculate a “supply factor” for each baseflow reservoir 1 & 2: If the demand is less than the storage of the actual reservoir, the supply factor is 1.0. Otherwise calculate a value between 0 and 1 = available storage / demand.



- 3 Calculate the final irrigation from each source of the actual Rank and Priority = calculated demand from actual baseflow reservoir 1 and/or 2 X corresponding supply factor.
- 4 Subtract the final irrigation volumes from the available volume of each baseflow reservoir 1 & 2 and go to next rank and priority.

In the next SZ Linear Reservoir time step, the calculated irrigation volumes are subtracted from the baseflow reservoir storages (and depths), and the irrigation pumping is stored together with the other SZLR results for water balance calculation, etc.

The available volume of water in each baseflow reservoir is calculated as

$$Available Volume_1 = \left(\begin{matrix} reservoir \\ depth \end{matrix} - \begin{matrix} depth \\ to \\ water \end{matrix} \right) \cdot \left(\begin{matrix} specific \\ yield \end{matrix} \right) \cdot \left(\begin{matrix} total \\ area \end{matrix} \right) \cdot \left(\begin{matrix} Fraction \\ Percolation \\ to \\ Reservoir_1 \end{matrix} \right)$$

$$Available Volume_2 = \left(\begin{matrix} reservoir \\ depth \end{matrix} - \begin{matrix} depth \\ to \\ water \end{matrix} \right) \cdot \left(\begin{matrix} specific \\ yield \end{matrix} \right) \cdot \left(\begin{matrix} total \\ area \end{matrix} \right) \cdot \left(1 - \begin{matrix} Fraction \\ Percolation \\ to \\ Reservoir_1 \end{matrix} \right)$$

Water Source Types

The Sources table specifies the different sources available for irrigation for a single command area. The order of the sources in the table is also their priority. For example, in the figure above, as long as there is sufficient water in the river, the irrigation water will be removed from the river. If the River falls below a specified level and/or discharge, then the irrigation water will be taken from the single well, and so on.



River Source

River source			
River Name:	<input checked="" type="checkbox"/> Use threshold river discharge		
<input type="text" value="Speed River"/>	Stop	Restart	
	River discharge:	<input type="text" value="2"/>	<input type="text" value="2.25"/> [m³/s]
River Chainage:	<input checked="" type="checkbox"/> Use threshold river stage		
<input type="text" value="32456"/> [m]	Stop	Restart	
Max rate:	<input type="text" value="0.5"/> [m³/s]	River stage:	<input type="text" value="230"/> <input type="text" value="230.25"/> [m]

To use a MIKE 11 river as a source of water, you must specify the MIKE 11 location to be used followed by the permitted river conditions that allow water to be removed. The River source actually has two conditions that can be used alone or combined.

River Name - The MIKE 11 Branch name of the river source. This name must exist in the MIKE 11 model and it must be spelled correctly.

River Chainage - The MIKE 11 chainage location to use for the river source. The model will find the nearest MIKE 11 H-point and use this for the river source.

Max rate - This is the maximum extraction rate for the river. If more water is required for irrigation, then the next source will be activated.

Use threshold river discharge - If the flow rate in the river falls below the **Stop** value, then water will no longer be taken from the River. However, if the flow rate in the river increases again and reaches the **Restart** value, the river source will be reactivated.

Use threshold river stage - If the water level in the river falls below the **Stop** value, then water will no longer be extracted from the River. However, if the water level in the river increases again and reaches the **Restart** value, the river source will be reactivated.

If both threshold values are specified, then the most critical one is used, and the source will not restart until both are satisfied.

Note There is no restriction on the number of river sources at a location. However, if the sources are located in the same model grid then a warning message will be printed to the *projectname_preprocessor_messages.log* file. The sources will be merged, retaining the maximum threshold stages and the sum of the capacities. The preprocessor also checks the license application volume to make sure these are the same. If not, the preprocessor will stop with an error.



Single Well Source

Single well source		
X-Pos:	Max depth to water:	Top of screen:
341290 [m]	8 [m]	0 [m]
Y-Pos:	Max rate:	Bottom of screen:
563210 [m]	0.35 [m ³ /s]	16 [m]

To use a well source in the model, you must specify the location and filter depth of the well. In a future release, this dialogue will be connected to the well database, but at the moment it is not.

X, Y -Pos - This is the X and Y map coordinates of the source well.

Max depth to water - this is the threshold value for the water depth in the well. If the water level in the well falls below this depth (as measured from the topography), the extraction will stop until the water level rises above the threshold.

Max rate - This is the maximum extraction rate for the well. If more water is required for irrigation, then the next source will be activated.

Top/Bottom of Screen - The depth of the top and bottom of the screen is used to define from which numerical layers water can be extracted. Pumping will stop if the water table falls below the bottom of the layer that contains the filter bottom.

There is no restriction on the number of wells at a location. However, if the wells are located in the same model grid, and have overlapping screen intervals, then a warning message will be printed to the *projectname_preprocessor_messages.log* file. The sources will be merged, retaining the maximum threshold depth, the sum of the capacities and the joint screening interval. The preprocessor also checks the license application volume to make sure these are the same. If not, the preprocessor will stop with an error.

In the linear reservoir groundwater method, multiple single wells are allowed in each baseflow reservoir. No warnings are given.

When the linear reservoir method is used, the screen interval is ignored and the water is pumped from the two baseflow reservoirs. The distribution between the two reservoirs is determined by the fraction given in the Baseflow Reservoirs (V.2 p. 106) dialogue. If the demand from one of the



reservoirs exceeds the available water, the pumping will be reduced. The pumping rate at the other reservoir will not be increased to compensate.

Also in the Linear Reservoir method, the specified “max depth to water” for the actual command area and source is used. In other words, it is not using the “threshold depth for pumping” in the baseflow reservoir menu. Pumping is allowed when the depth to the water table is less than the specified threshold value at the start of the time step.

Shallow Well Source

Shallow well source

Max depth to water:	<input type="text" value="8"/> [m]	Top of screen:	<input type="text" value="0"/> [m]
Max rate:	<input type="text" value="0.25"/> [m³/s]	Bottom of Screen:	<input type="text" value="16.25"/> [m]

In many cases, farmers have several shallow wells for irrigation, most of which may not be mapped exactly. Especially in regional scale models, each grid cell could thus contain many shallow groundwater wells. In such cases, the Shallow Well source can be used to simply extract water for irrigation from the same cell where it is used, without having to know the exact coordinates of the wells. By specifying this option, one well is placed in each cell of the command area.

Note: A cell (i, j, layer) containing a shallow well cannot also have a single well specified in the same cell (i.e. the same cell and layer).

Max depth to water - this is the threshold value for the water depth in the well. If the water level in the well falls below this depth (as measured from the topography), the pumping will stop until the water rises above the threshold depth again.

Max rate - This is the maximum extraction rate for the shallow well in each cell. If more water is required for irrigation, then the next source will be activated.

Top/Bottom of Screen - The depth of the top and bottom of the screen is used to define from which numerical layers water can be extracted. Pumping will stop if the water table falls below the bottom of the layer that contains the filter bottom.

Shallow well sources are removed from baseflow Reservoir 1 if the Linear Reservoir groundwater method is used. The screen interval is ignored.



Note Shallow wells can be located in cells containing single sources. The preprocessor will give a warning for such violations. Multiple shallow wells are not allowed in the same command area.

External Source

In some case, the irrigation water can be from outside of the watershed being modelled. In this case, the only constraint is the maximum amount of water than can be extracted from the source.

Max rate - This is the maximum extraction rate for the source. If more water is required for irrigation, then the next source will be activated.

Water Application methods

There are three ways to apply the irrigation water in the model.

Sprinkler - If the water is applied as sprinkler irrigation, it is added to the precipitation component.

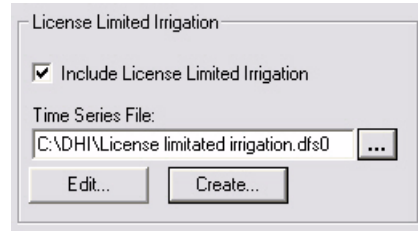
Drip - If the water is applied as Drip irrigation, it is added directly to the ground surface as ponded water.

Sheet - If the water is applied as Sheet irrigation, then an additional data tree item is required to define where the water is to be added within the command area. The idea behind this option is that water is flooded onto one or more cells of the command area and then distributed to the adjoining cells as overland flow. The sheet irrigation is applied directly to the cells as ponded water.

All three methods are allowed in the Simple, sub-catchment based overland flow method. However, the sheet method does not really make sense if the subcatchment overland flow method is used.



License Limited Irrigation



Sometimes, the total amount of irrigation water that a user can apply is limited by a license over a certain period (e.g. 10000 m³ / year). The license limited option, allows you to specify a dfs0 time series file with a time series of maximum amounts. If the maximum amount is reached within the license period, then the irrigation will be stopped until the next license period, when it will be started again. The license period length is defined by the time steps in the specified dfs0 file.

During the simulation the license data is included in the calculation of the available water volume of each source. The module keeps track of the “actual available license volume”. Whenever this is reached or exceeded, the source will be closed until a new license period starts (or the end of the simulation). When a source is closed for this reason, a message is printed in the wm_print.log file.

Notes

- The dfs0 file EUM Data Units (*V.1 p. 271*) must be Water volume (m³ or other volume unit) and the time series-type must be Step Accumulated (*V.1 p. 247*).
- The specified volumes cover the period from the previous value (or start of simulation) until the date of the actual value.
- The files may contain delete values. These are simply ignored. This makes it possible to include licenses for several sources in one file, even when the dates of the different source licenses differ.
- A new irrigation log file has been included in the results output: *projectname_IrrigationLicenseLog.dfs0*.
- This log file contains the “actual available license volume” of each source with license included, stored as instantaneous values at the end of every time step. This makes it easy to identify the periods where sources have been closed due to “license shortage”.
- **Note** Unused license volumes are NOT carried over to the next license period (use it or loose it !).



2.10.4 Sheet Application Area

Sheet Application Area

Conditions:	Irrigation selected in Land Use and UZ and ET simulated and Sheet selected as an application method
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code

The sheet application area is used to define where the sheet irrigation must be applied. The program does not make any distinction between sheet application areas. The sheet irrigation will be distributed on every cell with a non-delete value or non-zero integer code within the command area.

2.10.5 Irrigation Demand

Irrigation demand

Demand type: Ref. moisture content:

ID: Temporal Distribution:

Moisture deficit start:

Moisture deficit end:

Irrigation Demand

Conditions:	Irrigation selected in Land Use and UZ and ET simulated
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The Irrigation Demand is used to describe when the water will be applied in the model.

The Irrigation Demand data item is divided into two dialogues. The first is the distribution dialogue for the Demand Areas and the second contains the information on when the water will be applied in each Demand area.



Demand Type

User Specified - For the User Specified demand type, the demand is not calculated. Rather it is simply specified as a constant value or as a time series.

Crop stress factor - The crop stress factor is the minimum allowed fraction of the crop specific reference ET that the actual ET is allowed to drop to before irrigation starts. That is, the minimum allowed $(Actual\ ET)/(Reference\ ET \times K_c)$ relationship. This should be a value between 1 and 0. If the actual Crop Stress factor falls below the given value, irrigation will be added.

Ponding depth - When using this option, the demand will be equal to the difference between the actual ponding depth and specified ponding depth. The option is typically used for modelling irrigation of paddy rice. If the ponding depth falls below the specified value then more irrigation water is added.

Max allowed deficit - The available water for crop transpiration (AW) is the difference between the actual water content and the water content at the wilting point for the root zone. The maximum available water for crop transpiration (MAW) is then the value of AW for the reference moisture content, where either the saturated water content or the field capacity can be specified for the reference moisture content. The deficit can then be defined as the fraction of the MAW that is missing and is a value between 0 and 1, where 0 is the deficit when the actual moisture content is equal to the reference moisture content, and 1 is the deficit when the available water for crop transpiration is zero, which is when the water content drops below the water content at the wilting point. When using the Max Allowed Deficit method, irrigation is started when the deficit exceeds the moisture deficit start value and stops at the moisture deficit end value.

If, for example, the reference moisture content is the field capacity and irrigation should start when 60 % of the maximum available water in the root zone is used and cease when field capacity is reached again, the value for the Start should be 0.6, and the value for the Stop value should be 0

Reference Moisture Content

If the Maximum allowed deficit is used, then the reference moisture content can be either for calculating the deficit is either the saturated water content or the field capacity water content.



Temporal Distribution

In each of the demand types the demand factor can be specified as a constant value or as a time series. However, an additional option in this combobox is to use the Vegetation Properties file. To use this option, you must include a vegetation properties file when you specify the vegetation. Further, you must specify the irrigation properties in the vegetation properties file itself. In this case, the Vegetation properties file, will contain all of the values needed by each of the different methods and the demand values cannot be input in this dialogue.

2.10.6 Irrigation Priorities

Irrigation Priorities	
Conditions:	Irrigation selected in Land Use and UZ and ET simulated and Priorities option selected in Land Use dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code

If there is insufficient water available to satisfy all of the irrigation demand, then the irrigation areas can be prioritised. In this case, each area of the model can be assigned a priority number (1 = highest priority). All the areas with the highest priority will be irrigated first. If there is sufficient water after the first areas have been irrigated, then the areas with the next highest priority will be irrigated.

However, if there is insufficient water to completely satisfy the demand of a particular priority region (all of the cells with priority value = 1, for example), then the water will be distributed to each of the cells based on either Equal Volume or Equal Shortage. The choice of the two priority schemes is assigned in the main **Land Use (V.2 p. 62)** dialogue.

Equal Volume - If the water is to be distributed based on equal volume, then all cells with the same priority number will receive an equal amount of water, regardless of their actual demand. For example, if there is a demand for 100 m³ of water in 10 cells, but only 50 m³ is available, then each cell will receive 5 m³ of water, regardless of the actual demand.

Equal Shortage - If the water is to be distributed based on equal shortage, then all cells with the same priority number will receive an amount of water that satisfies an equal percentage of their actual demand. For



example, if there is a demand for 100 m³ of water in 10 cells, but only 50 m³ is available, and 5 of the cells have a demand of 15 m³, while 5 have a demand of only 5 m³, then the first cells will receive 7.5 m³, while the latter will receive only 2.5 m³.

2.11 *Evapotranspiration*

There are no options on the Evapotranspiration main dialogue and only one item in the data tree - reference evapotranspiration.

2.11.1 *Reference Evapotranspiration*

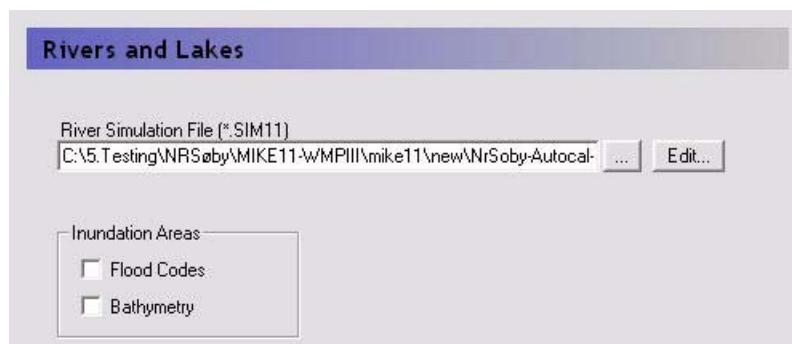
Reference Evapotranspiration	
dialogue Type	Time-varying Real Data
EUM Data	Grid Code
Units	
Time Series	Evapotranspiration Rate
EUM Data	
Units	

The reference evapotranspiration (ET) is the rate of ET from a reference surface with an unlimited amount of water. Based on the FAO guidelines, the reference surface is a hypothetical grass surface with specific characteristics. The reference ET value is independent of everything but climate and can be calculated from weather data. The FAO Penman-Monteith method is recommended for determining the reference ET value.

The Reference Evapotranspiration item comprises both a distribution and a value. The distribution can be either uniform, station-based or fully distributed. If the data is station-based then for each station a sub-item will appear where you can enter the time series of values for the station.



2.12 Rivers and Lakes



In this dialogue, you can link MIKE SHE to a MIKE 11 simulation (.sim11) file. The simulation file is the main MIKE 11 file, which contains the file references to the MIKE 11 river network (.nw11), cross-section (.xns11) and boundary (.bnd11) files.

In principle, there are three basic steps for developing an integrated MIKE SHE/MIKE 11 model:

- 1 Establish a MIKE 11 HD hydraulic model as a stand-alone model, make a performance test and, if possible, a rough calibration using prescribed inflow and stage boundaries.
- 2 Establish a MIKE SHE model that includes the overland flow component and (optionally) the saturated zone and unsaturated zone components.
- 3 Couple MIKE SHE and MIKE 11 by defining branches (reaches) where MIKE 11 should interact with MIKE SHE.

The chapter *Coupling MIKE 11 and MIKE SHE (V.1 p. 165)* describes in more detail the three steps above.

There are two additional options in the above dialogue for calculating inundation areas:

Flood codes - Flood codes are used for the Flood Area Option (*V.1 p. 174*) in MIKE 11 to indicate which cells flood during a storm event.

Bathymetry - The bathymetry is used to more accurately simulate the topography of the flood code cells, when the MIKE SHE topography is specified on a larger grid.

Related Items:

- Channel Flow - Reference (*V.2 p. 227*)



- Coupling of MIKE SHE and MIKE 11 (*V.2 p. 228*)
- River-Aquifer Exchange (line source/link) (*V.2 p. 234*)
- Area Inundation using Flood Codes (areal source/sink) (*V.2 p. 239*)
- Direct Overbank Spilling to and from MIKE 11 (*V.2 p. 240*)
- Coupling MIKE 11 and MIKE SHE (*V.1 p. 165*)
- MIKE 11 Cross-sections (*V.1 p. 166*)
- MIKE SHE Coupling Reaches (*V.1 p. 169*)

2.12.1 Flood codes

Flood Codes	
Conditions:	If Rivers and Lakes selected in the Simulation Specification dialogue and Flood Codes selected in the River and Lakes dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code

Flood codes are required when coupling MIKE 11 with the Inundation options by Flood Code. This requires a .dfs2 file with Integer Grid Codes which are then used for making the flood mapping for the coupling reaches.

Related Items:

- Area Inundation using Flood Codes (areal source/sink) (*V.2 p. 239*)
- Flood Area Option (*V.1 p. 174*)
- Inundation options by Flood Code (*V.1 p. 173*)



2.12.2 Bathymetry

Bathymetry	
Conditions	If Rivers and Lakes selected in the Simulation Specification dialogue and Bathymetry selected in the Rivers and Lakes dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Elevation

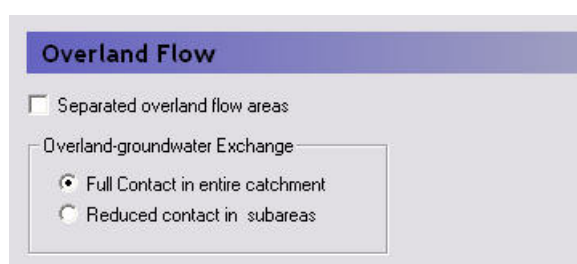
The bathymetry option allows you to specify a detailed flood plain and river bottom topography, which can then be used for more accurate definition of the topography in the flood code cells. This file is only used if the Bed Topography Option is set in MIKE 11.

Related Items:

- Area Inundation using Flood Codes (areal source/sink) (*V.2 p. 239*)
- Inundation options by Flood Code (*V.1 p. 173*)

2.13 Overland Flow

The main dialogue for overland flow includes several options when the Finite difference method is selected. It does not contain any options when the Sub-catchment based method is selected.



If the Finite Difference method is selected in the Simulation Specification dialogue, the basic items required for the calculation of Overland Flow are

- the Manning number, which is equivalent to the Stickler roughness coefficient,
- the Detention Storage, and



- the Initial Water Depth on the ground surface (ponded water).

There are two options available in the Overland Flow dialogue.

Separated overland flow areas The first allows you to divide the model area into overland flow zones, which are conceptually areas separated by dikes or embankments. With this option, overland flow will not be allowed to flow between zones. If this is checked, then an additional item, Separated Flow Areas (V.2 p. 87), will be added to the data tree.

Overland-groundwater exchange If the soil profile becomes completely saturated, then the unsaturated flow component, which normally controls surface-subsurface exchange, is no longer active. In this case, the overland flow module must exchange water with the saturated zone component directly. The Overland-groundwater exchange option allows you to specify an exchange coefficient to reduce the exchange of water between the overland flow and the saturated zone. If the reduced contact in areas is chosen then a new item, the Overland-groundwater Leakage Coefficient (V.2 p. 85), must be specified.

Related Items:

- Separated Flow Areas (V.2 p. 87)
- Overland-groundwater Leakage Coefficient (V.2 p. 85)
- Overland Flow - Reference (V.2 p. 211)
- Adding Overland Flow (V.1 p. 46)
- Coupling MIKE 11 and MIKE SHE (V.1 p. 165)

2.13.1 Manning number

Manning number	
Condition	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue
dialogue Type	Stationary Real Data
EUM Data	ManningsM
Units	

The Manning M is equivalent to the Stickler roughness coefficient, the use of which is described in Overland Flow - Reference (V.2 p. 211).



The Manning M is the inverse of the more conventional Mannings n. The value of n is typically in the range of 0.01 (smooth channels) to 0.10 (thickly vegetated channels). This corresponds to values of M between 100 and 10, respectively. Generally, lower values of Mannings M are used for overland flow compared to channel flow.

Related Items:

- Adding Overland Flow (*V.1 p. 46*)
- Overland Flow - Reference (*V.2 p. 211*)

2.13.2 Detention Storage**Detention Storage**

Conditions	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue
dialogue Type	Stationary Real Data
EUM Data	Storage Depth
Units	

Detention Storage is used to limit the amount of water that can flow over the ground surface. The depth of ponded water must exceed the detention storage before water will flow as sheet flow to the adjacent cell. For example, if the detention storage is set equal to 2mm, then the depth of water on the surface must exceed 2mm before it will be able to flow as overland flow. This is equivalent to the trapping of surface water in small ponds or depressions within a grid cell.

Water trapped in detention storage continues to be available for infiltration to the unsaturated zone and to evapotranspiration. Using detention storage, you can simulate water that is trapped in depressions that are smaller than a grid cell.

Related Items:

- Adding Overland Flow (*V.1 p. 46*)
- Overland Flow - Reference (*V.2 p. 211*)



2.13.3 Initial Water Depth

Initial Water Depth	
Condition	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Water Depth

This is the initial condition for the overland flow calculations, that is the initial depth of water on the ground surface. The initial water depth is usually zero.

The initial water depth for overland flow is also the boundary condition for overland flow. In other words, if you specify an initial depth of 2mm, then the boundary will always have 2mm of water and there will be an inflow of water to the model whenever the internal cell has less than 2mm of ponded water.

Related Items:

- Adding Overland Flow (*V.1 p. 46*)
- Overland Flow - Reference (*V.2 p. 211*)

2.13.4 Overland-groundwater Leakage Coefficient

Overland-groundwater Leakage Coefficient	
Conditions	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue AND the Reduced contact in subareas option selected in the main Overland Flow dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Leakage Coeff./Drain Time Const.

When the soil profile becomes completely saturated, MIKE SHE disables the unsaturated zone calculation. If, at the same time, there is ponded water on the ground surface, the exchange of water between the overland flow component and the groundwater component is calculated based on the vertical hydraulic conductivity in the upper layer of the saturated zone



and the hydraulic gradient between the surface water level and the ground water table in the upper layer of the saturated zone.

However, often the vertical hydraulic conductivity of the upper layer in the saturated zone does not represent the permeability of the top layer of the soil. The soil layers are usually described in more detailed in the UZ model than in the SZ model, but the UZ parameters are not used when UZ ‘disappears’.

To handle such situations, a leakage coefficient can be specified. The exchange of water between the surface water and ground water is then calculated based on the specified leakage coefficient and the hydraulic head between surface water and ground water. In other words the UZ model is automatically replaced by a simple Darcy flow description when the profile becomes completely saturated.

This option is often useful under lakes or on flood plains, which may be permanently or temporarily flooded, and where fine sediment may have accumulated creating a low permeable layer (lining) with considerable flow resistance. The value of the leakage coefficient may be found by model calibration, but a rough estimate can be made based on the saturated hydraulic conductivities of the unsaturated zone or in the low permeable sediment layer, if such data is available.

The specified leakage coefficient is used wherever it is specified. In areas where a delete value is specified, the vertical hydraulic conductivity of the top SZ layer is used.

In the processed data, the item, Overland-SZ Exchange Grid Code, is added, where areas with full contact are defined with a 0, and areas with reduced contact are defined with a 1.

Related Items:

- Inundation options by Flood Code (*V.1 p. 173*)
- Bed Leakage (*V.1 p. 176*)
- Overland Flow - Reference (*V.2 p. 211*)



2.13.5 Separated Flow Areas

Separated Flow Areas	
Conditions	when Overland Flow + the Finite Difference method is selected in the Simulation Specification dialogue and Separated Flow Areas selected in the main Overland Flow dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code

By specifying separated overland flow areas you can simulate areas that are separated by dikes or embankments. Separated overland flow areas are defined by specifying a .dfs2 Integer Grid Code file, containing a unique code value for each flow area. The model will then disable overland flow between grids with different flow codes. Thus, embankments can be simulated by defining different flow codes on each side of the embankment. Legal code values are 1 and higher. Delete values are not allowed inside the model area.

Related Items:

- Overland Flow - Reference (V.2 p. 211)

2.13.6 Overland Flow Zones

Topographical Routing

Name:

Slope: Slope length: [m]

Manning Number: [m^{1/3}/s] Retention storage: [mm]

Initial Depth: [m]

Overland Flow Zones	
Conditions:	If Overland Flow + the Subcatchment-based method is chosen in the Simulation Specification dialogue
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code



The Overland flow zones are used to define the topographic zones for the simple, catchment-based overland flow solution. The overland flow zones are typically defined as areas with similar topography. For example, the river flood plain would be a typical topographic zone, although it might be included in many subcatchments.

Each unique grid code in the Overland Flow Zones map will generate a sub-item in the data tree where the following parameters must be specified.

Slope - The Slope variable is a representative slope in the topographic area. It can be thought of as the average slope, but it is really a calibration parameter as it can't really be calibrated to a true average.

Slope Length - Like the Slope itself, the Slope Length is a representative distance that water must travel as overland flow before reaching a ditch or stream. It can be thought of as the average travel distance, but, like the Slope, it is really a calibration parameter as it can't really be calibrated to a true average.

Manning Number - The Manning M is equivalent to the Stickler roughness coefficient, the use of which is described in the Overland Flow - Reference chapter. The Manning M is the inverse of the more conventional Mannings n. The value of n is typically in the range of 0.01 (smooth channels) to 0.10 (thickly vegetated channels). This corresponds to values of M between 100 and 10, respectively. Generally, lower values of Mannings M are used for overland flow compared to channel flow.

Detention Storage - Detention Storage is used to limit the amount of water that can flow over the ground surface. For example, if the detention storage is set equal to 2mm, then the depth of water on the surface must exceed 2mm before it will be able to flow as overland flow. Water trapped in detention storage continues to be available for infiltration to the unsaturated zone and to evapotranspiration. Using detention storage, you can simulate water that is trapped in depressions that are smaller in area than a grid cell.

Initial Depth - This is the initial condition for the overland flow calculations, that is the initial depth of water on the ground surface.

Related Items:

- Simplified Overland Flow Routing (*V.2 p. 220*)
- Overland Flow - Reference (*V.2 p. 211*)



2.13.7 Dispersion coefficient along columns/rows

Dispersion coefficient along columns/rows	
Condition	when water quality for Overland Flow is selected in the Water Quality Simulation Specification dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Dispersion coefficient

For the overland flow transport, two dispersion coefficients are required - one along the rows and the other along the columns.

Note Unlike for the unsaturated and saturated flow, the overland transport module requires the actual dispersion coefficient [m^2/s] - not the dispersivity [m].

Related Items:

- Solute Transport in Overland Flow (V.2 p. 344)

2.13.8 Initial Mass (per Species)

Initial Mass	
Condition	when water quality for Overland Flow is selected in the Water Quality Simulation Specification dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Mass per unit area

The initial mass for overland transport is given as a surface concentration [e.g kg/m^2]. This makes it easier to control the mass of solute introduced because you do not have to consider surface water depth.

Related Items:

- Solute Transport in Overland Flow (V.2 p. 344)
- Initial Conditions (V.2 p. 347) in Overland Transport



2.14 Unsaturated Zone

Unsaturated Flow

Calculation Column Classification Type

- ☒ 1: Automatic
- ☐ 2: Specified calculation points
- ☐ 3: Calculated in all grid points
- ☐ 4: Partial automatic (combination of 1 and 2)

Bypass Flow

- ☒ No
- ☐ Yes

There are three methods in MIKE SHE to calculate Unsaturated Flow:

- Richards Equation,
- the Gravity Flow, and
- the Two-Layer Water Balance.

Both the Richards Equation method and the Gravity Flow use soil profiles that can have different soils at different depths. Whereas, the Two-Layer Water Balance uses a uniform soil for the entire depth. This leads to two distinct ways of entering UZ soil information in MIKE SHE - each with a distinct set of parameter dialogues:

- as a soil profile, or
- as a uniform soil.

If either the Richards Equation method or the Gravity Flow are used, then the data tree can include the following items:

- Soil Profile Definitions,
- Groundwater Depths used for UZ Classification,
- Partial automatic classification, and
- Specified classification.

Whereas, if the Two-Layer Water Balance is used then the data tree will include:



- 2-Layer UZ soil properties and
- ET Surface Depth.

Column Classification

If the either Richards Equation or the Gravity Flow Module are chosen for calculating the unsaturated zone flow then the top-level Unsaturated Zone dialogue contains a section for the column classification. However, if the Two-Layer Water Balance is chosen then the Column Classification section is hidden.

Since UZ computations in all grid squares for most large-scale applications requires excessive computation time, MIKE SHE enables you to compute the UZ flow in a reduced subset of grid squares. The subset classification is done automatically by the pre-processing program according to soil types, vegetation types, climatic zones, and depth to the groundwater table.

- **Automatic classification** The automatic classification requires a distribution of groundwater elevations (see Groundwater Depths used for UZ Classification). This can be either the initial depth to the groundwater based on the initial heads, or you can supply a .dfs2 map of the groundwater elevations. In both cases, you must supply a table of intervals upon which the classification will be based. The number of computational columns depends on how narrow the intervals are specified. If, for example, two depths are specified, say 1 m and 2 m, then the classification with respect to the depth to groundwater will be based on three intervals: Groundwater between 0 m and 1 m, between 1 m and 2 m, and deeper than 2 m.

If the Linear Reservoir method is used for the groundwater, then the Interflow reservoirs are also used in the classification. However, since feedback to the UZ only occurs in the lowest Interflow reservoir of each subcatchment, the Interflow reservoirs are added to the Automatic Classification in two zones - those that receive feedback and those that don't.

- **Specified classification** Alternatively a data file specifying Integer Grid Codes, where UZ computations are carried out can be specified, with grid codes range from 2 up to the number of UZ columns (see Specified classification). The location of the computational column is specified by a negative code and the simulation results are then transferred to all grids with the an equivalent positive code.



- **Calculated in all Grid points** For smaller scale studies, or studies where the classification system becomes intractable, you can specify that computations are to be carried out in all soil columns.
- **Partial Automatic** Finally a combination of the Automatic classification and the Specified classification is available. If this option is chosen an Integer Grid Code file must be provide (see Partial automatic classification) with the following grid codes: In grid points where automatic classification should be used the grid code 1 must be given. In grid points where computation should be performed for all cells the grid code 2 must be given.

Bypass Flow

Bypass flow is described in the Reference section under Simplified Macropore Flow (bypass flow). This flag simply determines whether it will be included or not.

If any soil zones include bypass flow, the bypass option should be checked.

Related Items:

- Unsaturated Flow - Reference (*V.2 p. 261*)
- Richards Equation (*V.2 p. 262*)
- Gravity Flow (*V.2 p. 273*)
- Two-Layer Water Balance (*V.2 p. 275*)
- Simplified Macropore Flow (bypass flow) (*V.2 p. 280*)
- Lumped UZ Calculations (*V.2 p. 281*)
- Coupling the Unsaturated Zone to the Saturated Zone (*V.2 p. 282*)



2.14.1 Soil Profile Definitions

UZ Soil Profile Definition

Profile ID:

Soil Profile:

	From depth	To depth	Soil name	UZ Soil property file		
1	0	0.5	A_ft	C:\5.Testing\...\DATABASE\Daisy_UzSoi	...	Edit...
2	0.5	2	C_ts	C:\5.Testing\...\DATABASE\Daisy_UzSoi	...	Edit...
3	2	10	C_dl	C:\5.Testing\...\DATABASE\Daisy_UzSoi	...	Edit...

Vertical Discretization:

	From depth	To depth	Cell height	No of cells
1	0	1	0.2	5
2	1	4	0.5	6
3	4	10	1	6

Bypass Const.

byp

thr1

thr2

Soil Profile Definitions

Conditions:	when Unsaturated Flow selected in the Simulation Specification dialogue and Richards Equation or the Gravity Flow module selected for the numeric engine
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The first part of the soil profile definition is to define the areas with the same soil profiles. Below this initial item a separate item will appear for every unique Grid Code in the file, in which the soil profile is defined (see below).

In this dialogue, the distribution of soil layers (i.e. depth and thickness of each soil type) in the individual profiles can be specified, as well as the vertical discretisation of the soil profile.

In the soil profile dialogue, there are four sections.



Header - The header includes the Profile ID, which is the editable name displayed in the data tree for this profile, and the Grid Code value, which is read from the Grid Code file.

Soil Profile - The soil profile section allows you to define the vertical soil profile. Soil layers can be added, deleted and moved up and down using the icons.

From and To Depths refers to the distances to the top and bottom of the soil layer, below the ground surface. Only the To Depth item is editable, as the From Depth item is equal to the bottom of the previous layer.

Soil name is the name of the soil selected in the UZ Soil Property file. It is not directly editable, but must be chosen from the list of available soil names when you assign the UZ Soil property file using the file browser.

UZ Soil property file is the file name of the soil database, in which the soil definition is available. The Edit button opens the specified Soil property database file, whereas the Browse button [...] opens the file browser to select a file.

Vertical discretisation - In this section you specify the vertical discretisation of the soil profile, which typically contains small cells near the ground surface and increasing cell thickness with depth. However, the soil properties are averaged if the cell boundaries and the soil boundaries do not align.

From and To Depths refers to the distances to the top and bottom of the soil layer, below the ground surface. Neither are directly editable since they are calculated from the number of cells and their thicknesses.

Cell Height is the thickness of the numerical cells in the soil profile.

No. of Cells is the number of cells with the specified cell height. Together these two values define the total thickness of the current section.

The discretisation should be tailored to the profile description and the required accuracy of the simulation. If the full Richards equation is used the vertical discretisation may vary from 1-5 cm in the uppermost grid points to 10-50 cm in the bottom of the profile. For the Gravity



Flow module, a coarser discretisation may be used. For example, 10-25 cm in the upper part of the soil profile and up to 50-100 cm in the lower part of the profile.

Note that at the boundary between two blocks with different cell heights, the two adjacent boundary cells are adjusted to give a smoother change in cell heights.

Bypass Constants The bypass parameters include:

byp - the maximum bypass fraction (between 0 and 1.0) of the net rainfall,

thr1 (θ_1) - the threshold water content below which the bypass fraction is reduced, and

thr2 (θ_2) - the minimum water content at which bypass occurs.

Related Items:

- Unsaturated Flow - Reference (V.2 p. 261)
- Richards Equation (V.2 p. 262)
- Gravity Flow (V.2 p. 273)
- Simplified Macropore Flow (bypass flow) (V.2 p. 280)



2.14.2 Groundwater Depths used for UZ Classification

	Depth
1	1
2	2.5
3	4
4	6

Water Table for Classification:

☐ Use initial water table for classification

☒ Specified water table for classification

Groundwater Depths used for UZ Classification

Conditions: if either the Automatic or Partially automatic column classifications selected in the main Unsaturated Zone dialogue

The automatic or partially automatic classification requires a distribution of groundwater elevations (see the main Unsaturated Zone dialogue). This can be either the initial depth to the groundwater based on the initial heads (default option), or you can supply a .dfs2 map of the groundwater elevations (second option). If you chose the second option, then a new item, Ground Water Table, will appear in the data tree.

In both cases, you must supply a table of intervals upon which the classification will be based. The number of computational columns depends on how narrow the intervals are specified. If, for example, two depths are specified, say 1 m and 2 m, then the classification with respect to the depth to groundwater will be based on three intervals: Groundwater between 0 m and 1 m, between 1 m and 2 m, and deeper than 2 m.

Related Items:

- Unsaturated Flow - Reference (V.2 p. 261)
- Lumped UZ Calculations (V.2 p. 281)



2.14.3 Ground Water Table

Groundwater Table	
Conditions:	if Specified water table for classification selected in the Groundwater Depths used for UZ Classification dialogue
dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Height above ground

If the Specified water table for classification is selected in the Groundwater Depths used for UZ Classification dialogue, then this is the ground water table used for the classification.

Related Items:

- Unsaturated Flow - Reference (V.2 p. 261)
- Lumped UZ Calculations (V.2 p. 281)

2.14.4 Partial automatic classification

Partial Automatic Classification	
Conditions:	if the Partially automatic column classifications selected in the main Unsaturated Zone dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code
Valid Values	1 or 2

A combination of the Automatic classification and the Specified classification is available. If this option is chosen an Integer Grid Code file must be provide with the following grid codes:

- In grid points where automatic classification should be used the grid code must be 1.
- In grid points where computation should be performed for all cells the grid code must be 2.

Related Items:

- Unsaturated Flow - Reference (V.2 p. 261)



- Lumped UZ Calculations (V.2 p. 281)

2.14.5 Specified classification

Specified Classification	
Conditions:	if Specified classification selected in the main Unsaturated Zone dialogue
dialogue Type	Integer Grid Codes
EUM Data Units	Grid Code
Valid values	+/- 2 to the number of SZ cells (-1, 0, 1 not valid)

This is a data file specifying Integer Grid Codes, where UZ computations are to be carried out. Grid codes range from 2, up to the number of UZ columns. The location of the computational column is specified by a negative code and the simulation results are then transferred to all grids with the an equivalent positive code.

For example, if a grid code holds the value -2, a UZ computation will be carried out for the profile located in that grid. Simulation results will subsequently be transferred to all grid codes with code value +2.

An easy way to generate a .dfs2 file to be used for specification of UZ computational columns is to let the MIKE SHE setup program generate an automatic classification first, and subsequently extract the UZ classification grid codes. The extracted .dfs2 file can be edited in the 2D editor as desired and used to specify UZ computational grids.

Related Items:

- Unsaturated Flow - Reference (V.2 p. 261)
- Lumped UZ Calculations (V.2 p. 281)



2.14.6 2-Layer UZ soil properties

2-Layer UZ properties	
Profile ID:	1: DLoam
Grid code value:	1
Soil water content at saturated conditions	0.4
Soil water content at field capacity	0.35
Soil water content at field wilting point	0.2
Infiltration rate	1e-006 [m/s]
Bypass const. byp 0.5 thr1 0.4 thr2 0.3	

2-Layer UZ Soil Properties

Conditions:	when Unsaturated Flow selected in the Simulation Specification dialogue and the Two-Layer Water Balance method selected for the numeric engine
dialogue Type:	Integer Grid Codes with sub-dialogue data
EUM Data Units	Grid Code

The first part of the soil profile definition is to define the areas with the same soil types. Below this initial item a separate item will appear for every unique Grid Code in the file, in which the soil characteristics are defined.

In this soil properties dialogue, there are three sections.

Header - The header includes the Profile ID, which is the editable name displayed in the data tree for this profile, and the Grid Code value, which is read from the Grid Code file.

Soil Properties In the Two-Layer Water Balance method, the soil database is not used. Instead, there are four principle parameters that must be defined for each soil type:

Soil water content at saturation - this is the maximum water content of the soil, which is usually approximately equal to the porosity,

Soil water content at field capacity - this is the water content at which vertical flow becomes negligible. In practice, this is the water content that is reached when the soil can freely drain. Although, it is usually higher than the residual saturation, which is usually defined as the minimum saturation that can be achieved in a laboratory test.



Soil water content at the field wilting point - this is the lowest water content that plants can extract water from the soil.

Infiltration rate - this is the saturated hydraulic conductivity of the soil.

Bypass Constants The bypass parameters include:

byp - the maximum bypass fraction (between 0 and 1.0) of the net rainfall,

thr1 (θ_1) - the threshold water content below which the bypass fraction is reduced, and

thr2 (θ_2) - the minimum water content at which bypass occurs.

Related Items:

- Unsaturated Flow - Reference (V.2 p. 261)
- Two-Layer Water Balance (V.2 p. 275)
- Simplified Macropore Flow (bypass flow) (V.2 p. 280)

2.14.7 ET Surface Depth

ET Surface Depth	
Conditions:	when Unsaturated Flow selected in the Simulation Specification dialogue and the Two-Layer Water Balance method selected for the numeric engine
dialogue Type	Stationary Real Data
EUM Data Units	Depth below ground

In unsaturated fine soils, capillary action can lead to saturated conditions existing some distance above the water table. If the water table is close to the ground surface, ET will continue to occur at the maximum rate, so long as this capillary zone reaches the ground surface. That is, evapotranspiration will not decrease the saturation, but draw water directly from the water table due to capillary action. Similarly, when the water table is deeper, plant roots will draw water directly from the saturated zone as long as the roots reach the capillary zone.

The ET surface depth equals the thickness of the capillary zone. It is used as the water table depth at which the ET starts to decrease. That is, if the



water table falls below the ET surface, then the linear function that reduces ET becomes active.

In coarse to medium sands, the ET surface depth is typically less than 10cm. In fine sands and silts, the ET surface depth could be a half a metre or more.

Note The ET surface depth must be greater than zero.

Related Items:

- Unsaturated Flow - Reference (V.2 p. 261)
- Two-Layer Water Balance (V.2 p. 275)
- Simplified ET for the Two-Layer Water Balance Method (V.2 p. 254)

2.15 Groundwater Table

Groundwater Table	
Conditions	If Unsaturated Flow selected in the Simulation Specification dialogue without selecting Saturated Flow, OR If the Unsaturated Flow is selected and the Linear Reservoir Method for groundwater is selected.
dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Height above ground

The groundwater table must be explicitly defined if unsaturated flow is simulated without explicitly simulating groundwater flow. The specified groundwater table is used as the lower boundary condition for the unsaturated model.

If the Linear Reservoir Method is used for the groundwater simulation, the water table is not calculated, thus requiring the water table to be explicitly defined.

However, the specified groundwater table is a static variable. If you need to relate your unsaturated zone model to a dynamic water table, you must include the saturated zone in your model based on the 3D Finite Difference Method.

**Related Items:**

- Saturated Flow - Reference (V.2 p. 289)
- Linear Reservoir Method (V.2 p. 307)

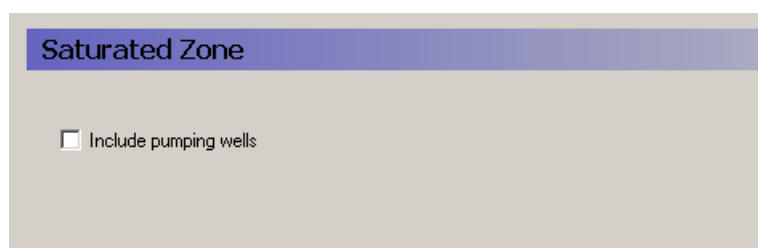
2.16 Saturated Zone

In MIKE SHE, the saturated zone is only one component of an integrated groundwater/surface water model. The saturated zone interacts with all of the other components - overland flow, unsaturated flow, channel flow, and evapotranspiration.

By comparison, MODFLOW only simulates the saturated flow. All of the other components are either ignored (e.g. overland flow) or are simple boundary conditions for the saturated zone (e.g. evapotranspiration).

On the other hand, there are very few difference between the MIKE SHE 3D Finite Difference Method numerical engine and MODFLOW. In fact, they share the same PCG solver. The differences that are present are limited to differences in the discretisation and to some differences in the way some of the boundary conditions are defined.

Linear Reservoir Method



Setting up a saturated groundwater model using the Linear Reservoir Method involves defining the Interflow and Baseflow Reservoirs, as well as their respective properties.

Pumping wells

By default, wells are not included, but in most applications pumping wells play a major role in the hydrology of the area. If wells are included in the model, then this must be checked and a new item in the data tree appears where the well database can be defined. Pumping wells extract water only from the baseflow reservoirs.

**Related Items:**

- Saturated Flow - Reference (V.2 p. 289)
- Linear Reservoir Method (V.2 p. 307)

3D Finite Difference Method

Setting up a saturated zone hydraulic model based on the 3D Finite Difference Method involves defining the:

- the geological model,
- the vertical numerical discretisation,
- the initial conditions, and
- the boundary conditions.

In the MIKE SHE GUI, the geological model and the vertical discretisation are essentially independent, while the initial conditions are defined as a property of the numerical layer. Similarly, subsurface boundary conditions are defined based on the numerical layers, while surface boundary conditions such as wells, drains and rivers (using MIKE 11) are defined independently of the subsurface numerical layers.

The use of grid independent geology and boundary conditions provides a great deal of flexibility in the development of the saturated zone model, thus the same geological model and many of the boundary conditions can be re-used for different model discretisation and different model areas.

The principle Saturated Zone dialogue for MIKE SHE includes three items.



Pumping wells

By default, wells are not included, but in most applications pumping wells play a major role in the hydrology of the area. If wells are included in the model, then this must be checked and a new item in the data tree appears where the well database can be defined.

Subsurface drainage

Subsurface drainage is used to limit the amount of groundwater that reaches the ground surface and to route near surface groundwater to local streams and rivers. There are a number of drainage options for specifying surface drains in MIKE SHE, which are described in more detail in the section Drainage.

Hydrogeologic parameter definition

The first option allows you to specify the hydrologic parameters of the geologic layers and lenses directly by means of .dfs2 grid files, point/line theme .shp files, or irregular xyz point values. The second option allows you to assign the hydrologic parameters to the geologic layers by means of zones with uniform properties, whereby the zones are defined by integer grid codes.

Dispersion

If your simulation includes water quality modelling in the saturated zone, then you must also define the type of dispersion you want to simulate. Dispersion is the physical process that causes solutes to spread longitudinally, vertically and horizontally as they move through the soil. The dispersion essentially represents the natural, microscopic variations in pore geometry that cause small scale variations in flow velocity.

No dispersion - Dispersion is ignored and no dispersivities need to be specified.

Isotropic - The transverse horizontal and transverse vertical dispersivities are assumed to be the same. Only two dispersivities need to be specified - the longitudinal and the transverse dispersivities.

Anisotropic - The horizontal and vertical transverse dispersivities are different, which requires the specification of five different dispersivities.

Related Items:

- Saturated Flow - Reference (*V.2 p. 289*)
- Solute Transport in the Saturated Zone (*V.2 p. 329*)



2.16.1 Interflow Reservoirs

The interflow reservoirs are used to route near-surface groundwater to local streams. In the Linear Reservoir Method, each reservoir is assumed to be like a bathtub, with an inflow from infiltration and the upstream reservoir, as well as an outflow flowing into the next downstream reservoir and down into the baseflow reservoir beneath. Each linear reservoir flows only into the next downstream interflow reservoir, or into a stream if it is the lowest reservoir.

Note Polygon shape files are currently not allowed for defining interflow reservoirs. The flow reference between interflow reservoirs depends precisely on the integer code numbers assigned to the reservoirs. Within a subcatchment, the interflow reservoir with the higher number always flows into the reservoir with the next lowest number.

Each Interflow reservoir requires a value for:

Specific Yield - to account for the fact that the reservoir contains a porous media, and is not an actual bathtub.

Initial depth - the initial depth of water in the reservoir, measured from the ground surface.

Bottom depth - the depth below the ground surface of the bottom of the reservoir. If the water level drops to the bottom of the reservoir, percolation stops.

Interflow time constant - a calibration parameter that represents the time it takes for water to flow through the reservoir to the next reservoir.

Percolation time constant - a calibration parameter that represents the time it takes for water to seep down into the baseflow reservoir



Interflow threshold depth - the depth below the ground surface when interflow stops. If interflow stops, percolation will continue until the reservoir is empty (i.e the water level reaches the bottom depth). The threshold depth must be less than or equal to the depth to the bottom of the reservoir.

Related Items:

- Saturated Flow - Reference (V.2 p. 289)
- Linear Reservoir Method (V.2 p. 307)
- Calculation of Interflow (V.2 p. 313)

2.16.2 Baseflow Reservoirs

The image shows two overlapping windows from the MIKE SHE software. The top window is titled 'Baseflow Reservoir' and contains the following fields: 'Name' (Global), 'Fraction of percolation to reservoir 1' (0), 'Fraction of pumping from reservoir 1' (0), and a checked checkbox for 'Use default river links'. The bottom window is titled 'Reservoir 1' and contains the following fields: 'Specific yield' (0), 'Time constant for base flow' (0 [d]), 'Dead storage fraction' (0), 'UZ feedback fraction' (0), 'Initial depth' (0 [m]), 'Threshold depth for base flow' (0 [m]), 'Threshold depth for pumping' (0 [m]), and 'Depth to the bottom of the reservoir' (0 [m]).

In the Linear Reservoir Method in MIKE SHE, each baseflow reservoir is divided into two parallel baseflow reservoirs. The two parallel baseflow reservoirs each receive a fraction of the percolation water from the interflow reservoirs as their only source of inflow. Each baseflow reservoir can discharge to pumping wells, to the unsaturated zone adjacent to streams and rivers (i.e. the zone beneath the lowest interflow reservoir), as well as directly to the MIKE 11 river network.



In the primary baseflow reservoir map view, you can define the number of baseflow reservoirs in your system. You can define any number of baseflow reservoirs, but typically, there are only one or two.

For each baseflow reservoir pair, there are three items to define:

Fraction of percolation to reservoir 1 - this is used to divide the percolation between each of the two parallel baseflow reservoirs.

Fraction of pumping from reservoir 1 - this is used to divide the pumping (if it exists) between each of the two parallel baseflow reservoirs.

Use default river links - in most cases you will link the simplified overland flow and the groundwater interflow to all of the river links found in the lowest interflow reservoir in each subcatchment. However, in some cases you may want to link the flow to particular river links. For example, if your MIKE 11 river network does not extend into the subcatchment, you can specify that the interflow discharges to a particular node or set of nodes in a nearby river network.

If you uncheck this checkbox, a River Links sub-item will appear where you can specify the river branch and chainage to link the subcatchment to.

In the sub-dialogue for each of the parallel baseflow reservoirs, you must define the following:

Specific Yield - to account for the fact that the reservoir contains a porous media and is not an actual bathtub.

Time constant for base flow - a calibration parameter that represents the time it takes for water to flow through the reservoir

Dead storage fraction - the fraction of the received percolation that is not added to the reservoir volume but is removed from the available storage in the reservoir.

UZ feedback fraction - the fraction of base flow to the river that is available to replenish the water deficit in the unsaturated zone adjacent to the river (i.e. the lowest interflow reservoir in the subcatchment).

Initial depth - the initial depth to the water in the reservoir measured from the ground surface



Threshold depth for base flow - the depth below the ground surface when base flow stops. The threshold depth must be less than or equal to the depth to the bottom of the reservoir.

Threshold depth for pumping - the depth below the ground surface when pumping is shut off. The threshold depth must be less than or equal to the depth to the bottom of the reservoir.

Depth of the bottom of the reservoir - the depth below the ground surface of the bottom of the reservoir.

Related Items:

- Saturated Flow - Reference (*V.2 p. 289*)
- Linear Reservoir Method (*V.2 p. 307*)
- Calculation of Baseflow (*V.2 p. 316*)

2.16.3 Geological Units

Geological units							
	Soil name	Soil code	Horizontal conductivity	Vertical conductivity	Specific yield	Storage coefficient	Porosity
1	DL	1	4e-006	2e-006	0.1	3e-005	0.6
2	DS+TS	10	5e-005	1e-005	0.15	3e-005	0.6
3	ES	3	5e-005	1e-005	0.15	3e-005	0.6
4	FT	4	4e-007	2e-007	0.15	3e-005	0.6
5	FP,FS,ML	8	4e-008	2e-008	0.1	3e-005	0.6
6	HG	6	2e-005	6e-006	0.15	3e-005	0.6
7	HS	7	1e-005	2e-006	0.15	3e-005	0.6
8	MS	9	2e-005	4e-006	0.15	3e-005	0.6

If you specify your geologic conceptual model via geological units, you can add each of your geologic units and its associated hydrogeologic properties to the table. Then, instead of specifying the hydrogeologic properties for each geological layer, you only need to specify the distribution of the units within the geologic layer or lense.

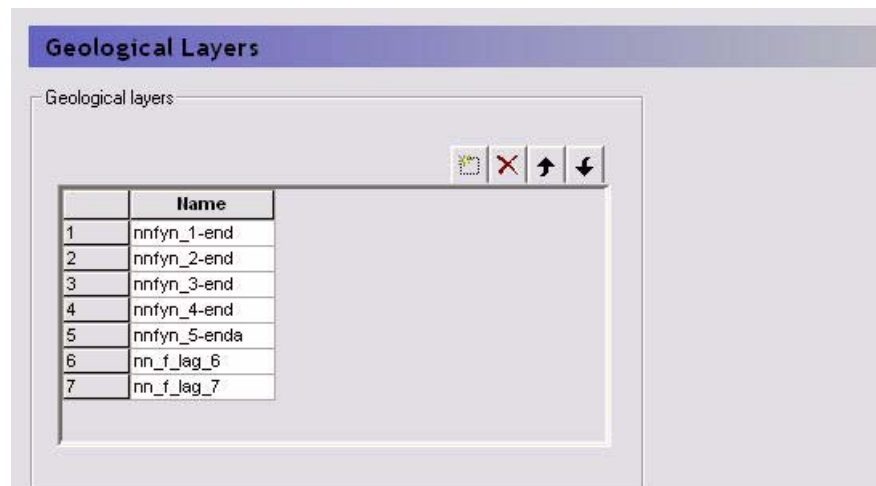
Related Items:

- Horizontal Hydraulic Conductivity (*V.2 p. 112*)
- Vertical Hydraulic Conductivity (*V.2 p. 113*)



- Specific Yield (V.2 p. 114)
- Specific Storage (V.2 p. 114)
- Porosity (V.2 p. 115)
- Dispersion Coefficients LHH, THH, TVH, LVV, THV (V.2 p. 115)

2.16.4 Geological Layers



For each geologic layer, you must specify the hydrogeologic parameters of the layer including

- Lower Level (Geological Layer or Lense, or Water Quality Layer),
- Horizontal Hydraulic Conductivity,
- Vertical Hydraulic Conductivity,
- Specific Yield,
- Specific Storage,

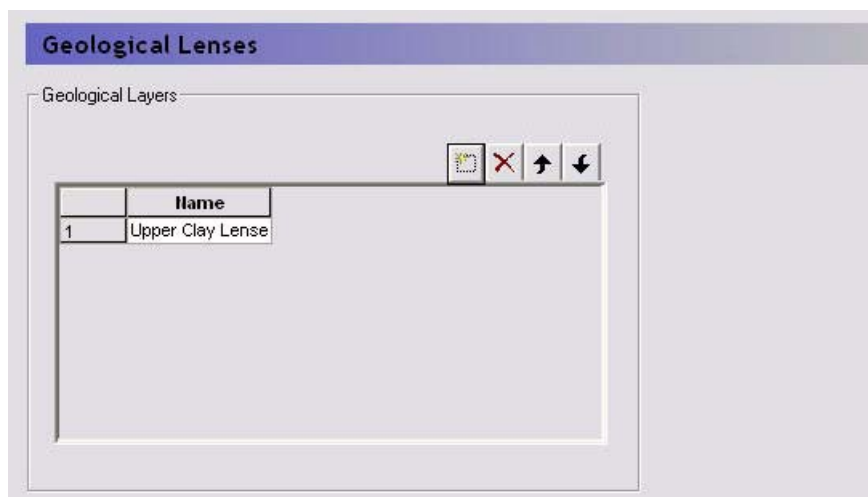
If you define your hydrogeology by , then most of the physical properties will be defined as properties of the Geological Unit and there will be an additional item, the Geological Unit Distribution, in the data tree.

Related Items:

- Saturated Flow - Reference (V.2 p. 289)



2.16.5 Geological Lenses



For each geologic layer, you must specify the hydrogeologic parameters of the layer including

- Lower Level (Geological Layer or Lense, or Water Quality Layer),
- Upper Level,
- Horizontal Extent,
- Horizontal Hydraulic Conductivity,
- Vertical Hydraulic Conductivity,
- Specific Yield,
- Specific Storage,

If you define your hydrogeology by , then most of the physical properties will be defined as properties of the Geological Unit and there will be an additional item, the Geological Unit Distribution, in the data tree.

Related Items:

- Working with Lenses (*V.I p. 52*)



2.16.6 Lower Level (Geological Layer or Lense, or Water Quality Layer)

Lower Level

dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Height above ground

The Lower Level value is used to define the bottom of geological layers geological lenses, and water quality layers. The bottom of geological or water quality layer is always equal to the top of the layer underneath. In the case of geological lenses, the lower level is used in the interpolation algorithm to interpolate geological properties to the model cells.

Related Items:

- Working with Lenses (*V.I p. 52*)

2.16.7 Upper Level

Upper Level

dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Height above ground

The Upper Level is the upper elevation of the lense. It is used by the interpolation algorithm to assign geological properties to the model cells.

Related Items:

- Working with Lenses (*V.I p. 52*)

2.16.8 Horizontal Extent

Horizontal Extent

dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code

The horizontal extent is used to define the lateral extents of geologic lenses. The horizontal extents is usually a .shp file polygon, or a dfs2 grid



file. In either case, the polygon name or the .dfs2 codes are ignored. Any cell within a polygon or with a grid code different than 0 is treated as part of the lense.

Related Items:

- Working with Lenses (*V.1 p. 52*)

2.16.9 Geological Unit Distribution

Geological Unit Distribution	
Conditions:	If geology defined by Geologic Units
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code
Valid Values	each value must be in the Geological Units table

The Geological Unit Distribution references the geological units defined in the table. Each Integer Code must refer to one of the geological units in the table.

Related Items:

- Geological Units (*V.2 p. 108*)
- Working with Lenses (*V.1 p. 52*)

2.16.10 Horizontal Hydraulic Conductivity

Horizontal Hydraulic Conductivity	
dialogue Type	Stationary Real Data
EUM Data Units	HydrConductivity

The hydraulic conductivity is a function of the soil texture and is related to the ease with which water can flow through the soil. Loose, coarse uniform soils have a higher conductivity than compacted soils with a range of particle sizes. Thus, a loose, uniform coarse sand can have a horizontal hydraulic conductivity as high as 0.001 m/s. Whereas, a tight, compacted clay can have a horizontal hydraulic conductivity as low as 1×10^{-8} m/s - which is 5 orders of magnitude.



The horizontal hydraulic conductivity is typically 5 to 10 times higher than the vertical hydraulic conductivity.

MIKE SHE assumes that the horizontal conductivity is isotropic in the x and y directions

Related Items:

- Geological Units (*V.2 p. 108*)
- Vertical Hydraulic Conductivity (*V.2 p. 113*)
- Working with Lenses (*V.1 p. 52*)

2.16.11 Vertical Hydraulic Conductivity

Vertical Hydraulic Conductivity	
dialogue Type	Stationary Real Data
EUM Data Units	HydrConductivity

The hydraulic conductivity is a function of the soil texture and is related to the ease with which water can flow through the soil. Loose, coarse uniform soils have a higher conductivity than compacted soils with a range of particle sizes. Thus, a loose, uniform coarse sand can have a horizontal hydraulic conductivity as high as 0.001 m/s. Whereas, a tight, compacted clay can have a horizontal hydraulic conductivity as low as 1×10^{-8} m/s - which is 5 orders of magnitude.

The vertical hydraulic conductivity is typically 5 to 10 times lower than the horizontal hydraulic conductivity.

Related Items:

- Geological Units (*V.2 p. 108*)
- Horizontal Hydraulic Conductivity (*V.2 p. 112*)
- Working with Lenses (*V.1 p. 52*)



2.16.12 Specific Yield

Specific Yield	
dialogue Type	Stationary Real Data
EUM Data Units	Specific Yield

In an unconfined aquifer, the Specific Yield is defined as the volume of water released per unit surface area of aquifer per unit decline in head. The specific yield is much higher than the Specific Storage because the water that is released is primarily from the dewatering of the pores at the water table. This results in a unit of $L^3/L^2/L$, which is dimensionless.

The Specific Yield is only used in transient simulations, but must always be input. Furthermore, the specific yield is only used in the cells that contain the water table. In the cells below the water table, the Specific Storage is used.

Related Items:

- Geological Units (*V.2 p. 108*)
- Working with Lenses (*V.1 p. 52*)

2.16.13 Specific Storage

Specific Storage	
dialogue Type	Stationary Real Data
EUM Data Units	Elastic Storage

In a confined aquifer, the specific storage is defined as the volume of water released per volume of aquifer per unit decline in head. This is slightly different than the specific yield because the water released from storage comes primarily from the expansion of the water and aquifer compression due to the reduction in water pressure (increase in effective stress). Thus, the water released from storage is released from the entire column of water in the aquifer, not just at the phreatic surface. This results in a unit of $L^3/L^3/L$, or $1/L$.

The Specific Storage Coefficient is only used in transient simulations, but must always be input. Furthermore, the specific storage coefficient is only



used in the cells below the water table. In the cells containing the water table, the Specific Yield is used.

2.16.14 Porosity

Porosity	
Conditions	if the Include Advection Dispersion (AD) Water Quality option selected in the Simulation Specification dialogue
dialogue Type:	Stationary Real Data
EUM Data Unitse:	Porosity

In a porous media, most of the volume is taken up by soil particles and the actual area available for flow is much less than the nominal area. This distinction is important when calculating flow velocities for solute transport. The porosity is the cross-sectional area available for flow divided by the nominal cross-sectional area. This is often referred to as the effective porosity, since it discounts the dead end pore spaces that are not available for flow. In the absences of dead end pores, the porosity is equal to the specific yield.

The Porosity must be greater than 0 and less than 1. For unconsolidated porous media, the porosity is usually from 0.15 to 0.3 depending of the grain size distribution (the more uniform the higher the effective porosity). For fractured media the porosity is usually much lower, in the from 0.01 to 0.05.

Related Items:

- Solute Transport in the Saturated Zone (V.2 p. 329)

2.16.15 Dispersion Coefficients LHH, THH, TVH, LVV, THV

Dispersion Coefficients LHH, THH, TVH, LVV, THV	
dialogue Type	Stationary Real Data
EUM Data Units	Dispersion Coefficient

If dispersion is included, then the two different dispersion options differ in the number of dispersion parameters required. If you assume isotropic conditions you need to specify the longitudinal dispersivity, α_L , and the



transversal dispersivity, α_T . If you assume anisotropic conditions you need to specify five dispersivities.

The magnitude of the dispersivity coefficient depends on the degree of heterogeneity in your geology and the degree to which these heterogeneities have been described in the model. The more heterogeneous your geology is, the larger the dispersivities should be. On the other hand, the more detailed you have described the heterogeneities with your model geometry, the smaller dispersivities should be.

Further, the magnitude of the dispersivities depends on the size of the model and on the model grid size. The larger the model, the larger the dispersivities should be. Whereas, the larger the grid size, the smaller the dispersivities should be due to numerical dispersion.

Thus, it is difficult to give a rule of thumb for the values of dispersivity. Recent field experiments on solute transport, though, indicate that the longitudinal dispersivity should be in the range of 1% or less of the travel distance, the transverse-horizontal dispersivity should be at least 50 times less than this and the transverse-vertical dispersivity should be 2 or more times less than the transverse-horizontal.

Related Items:

- Saturated Zone (V.2 p. 102)
- Solute Transport in the Saturated Zone (V.2 p. 329)

2.16.16 Computational Layers

I	Name
1	Aquifer-top half
2	Aquifer-bottom half
3	Silt
4	Limestone

The vertical discretisation in the saturated zone can be defined in two ways:



- by the geological layers, in which case there will be one calculation node in each geological layer,
- by explicitly defining the lower level of each calculation layer.

Vertical discretisation

Defined by the geological layers - Groundwater flow in a multi-layer aquifer can be described by a model in which the computational layers follow the interpreted geological layers. Each layer is characterised only by its base level specified either by a constant level or by a distributed file. The number of numerical layers will be identical with the number of geologic layers.

Explicit definition of lower levels - If you define the vertical discretisation explicitly each computational layer is defined by its lower elevation.

Minimum layer thickness - The minimum thickness of the calculation layers must be specified to adjust the geological model or the specified levels to prevent layers with zero thickness.

2.16.17 Lower Level (Numerical Layer)

Lower Level	
dialogue Type	Stationary Real Data
EUM Data Units	Elevation

The Lower Level value is used to define the bottom of the numerical layers, if they are being explicitly defined. The bottom of the numerical layers is always equal to the top of the layer underneath.

2.16.18 Initial Potential Head

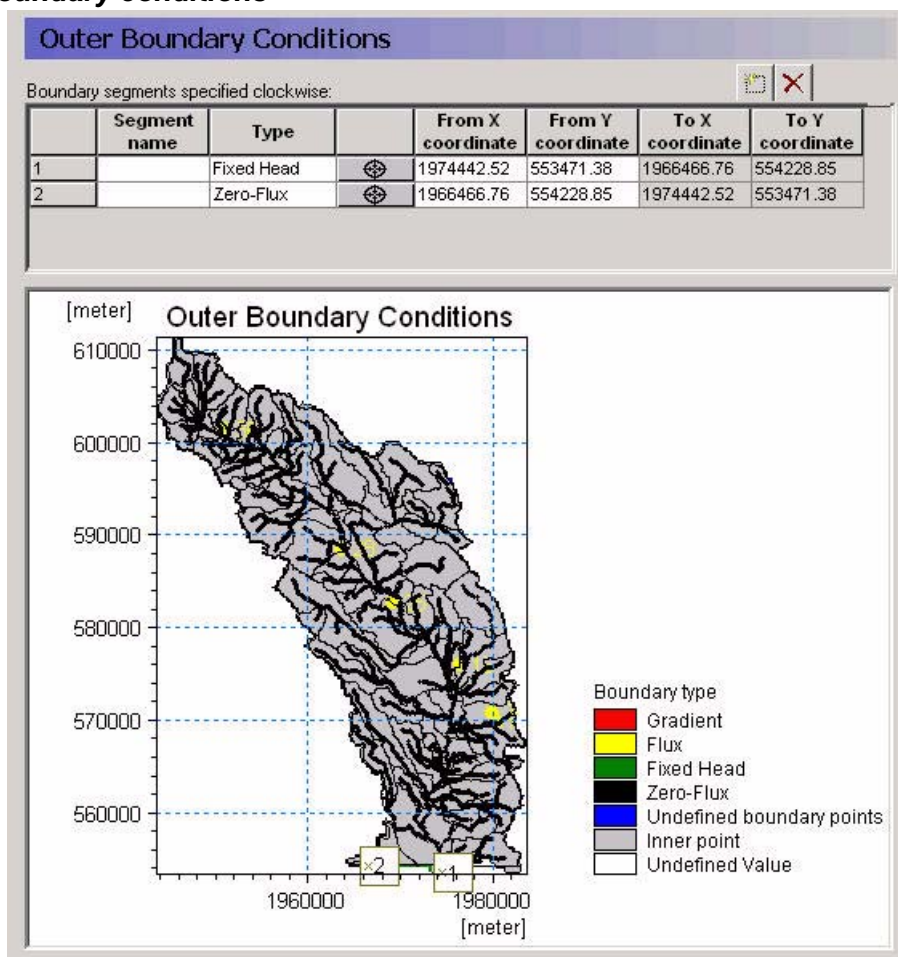
Initial Potential Head	
dialogue Type	Stationary Real Data
EUM Data Units	Elevation

The Initial Potential Head is the starting head for transient simulations and the initial guess for steady-state simulations. The choice of initial head for



steady state simulations may affect the rate of numerical convergence depending on the solver used.

2.16.19 Outer boundary conditions



The outer boundary conditions are defined as line segments between two boundary points. The boundary points are, in principle, independent of the model domain because they do not need to lie on the model boundary. Rather they are projected onto the nearest model boundary cell. Thus, the model boundary can be modified slightly without having to modify the boundary locations. However, if the model boundary is moved significantly or if the boundary is convoluted then the calculated 'nearest' node might not be the one expected.

Specifying a boundary condition

To specify an outer boundary segment,



- 1 add a new line to the outer boundary points table
- 2 click on the target icon,
- 3 click on one end of your boundary segment,
- 4 add a second line to the outer boundary points table
- 5 click on the new target icon,
- 6 click on the other end of your boundary segment
- 7 Change the name of the top line of the points table,
- 8 Select the appropriate boundary condition for the boundary segment.

Available boundary conditions

Fixed Head - This boundary prescribes a head in the boundary cell. The head can be fixed at a prescribed value, fixed at the initial value from the initial conditions, or assigned to a .dfs0 time series file.

The last option is a time varying dfs2 file, which is typically extracted from a regional results file. This can be done using the MIKE Zero Toolbox Extraction tool: 2D Grid from 3D files. MIKE SHE then interpolates in both time and space from the .dfs2 file to the local head boundary at each local time step.

Zero flux - This is a no-flow boundary, which is the default.

Flux - This boundary describes a constant or time varying flux across the outer boundary of the model. A time varying flux can be specified as a mean step-accumulated discharge (e.g. m^3/s) or as a step-accumulated volume (e.g. m^3). A positive value implies an inflow to the model cells.

Gradient - This boundary describes a constant or time varying gradient between the node on the outer boundary and the first internal node. A time varying gradient can be specified as an instantaneous dimensionless or percent value. A positive gradient implies a flux into the model.

Notes

- 1 The head is calculated in a No Flow outer boundary cells, whereas the head is specified in the Fixed Head outer boundary cells, but in both cases all properties must be assigned to all outer boundary cells.
- 2 An internal model cell in contact with multiple boundary cells will not receive multiple quantities of water.



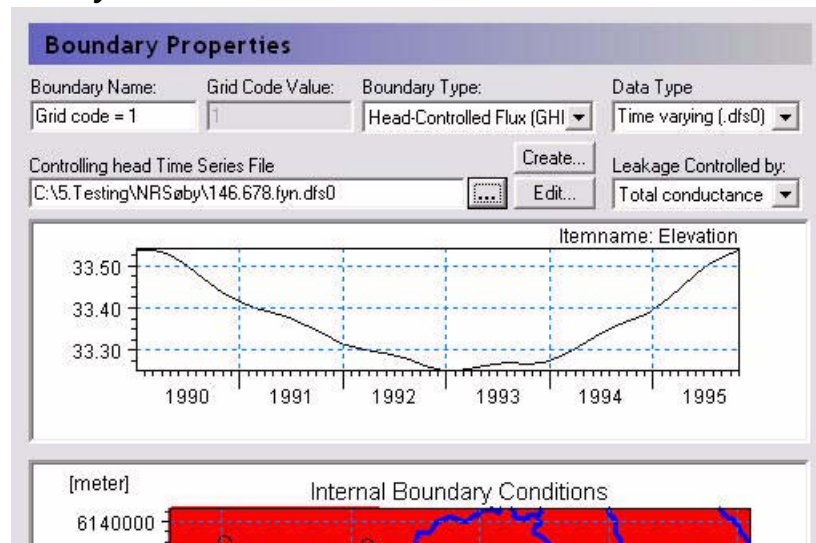
- 3 Additional detailed information can be found under Boundary Conditions (*V.2 p. 296*) in the Technical Reference for the Saturated Zone module.
- 4 An error will be generated if the flux/gradient input cell
 - has zero thickness,
 - has a horizontal hydraulic conductivity of zero,
 - is an inactive internal boundary cell, or
 - is a fixed head internal boundary cell.
- 5 A warning will be issued if the flux/gradient boundary
 - is a head controlled flux (GHB) internal boundary cell, or
 - is a fixed head drain internal boundary cell.

Related Items:

- Saturated Flow - Reference (*V.2 p. 289*)
- Boundary Conditions (*V.2 p. 296*)



2.16.20 Internal boundary conditions



Internal Boundary Conditions

dialogue Type: Integer Grid Codes with sub-dialogue data

EUM Data Units Grid Code

The map dialogue for the internal boundary conditions allows you to specify the locations of the various boundary conditions. Any Integer Code value is permissible and a separate item will be added to the data tree below this level with one item for each unique integer code in the domain.

Only one integer code is allowed per cell, which means that no cell can have more than one boundary condition.

If you use a polygon .shp file, you can create and edit the polygons in the Simple Shape Editor (V.2 p. 201). Each unique polygon will have a separate entry in the data tree.

Available boundary conditions

The following boundary conditions can be defined for each integer code:

Fixed Head For a fixed head boundary, you specify the head in the cell. The model will not calculate the head in the cell. Care should be taken when specifying fixed head boundary conditions, as the cell becomes an infinite source or sink of water. The fixed head can be a prescribed value, fixed at the initial value from the initial conditions, assigned to a .dfs0 time series file, or assigned to a time varying dfs2 file. The last



option is typically from a results file. It could be from a regional results file, which can be extracted using the MIKE Zero Toolbox Extraction tool: 2D Grid from 3D files. Or, it could be from a previous run of the same model.

Fixed Head Drain - For a fixed head drain boundary, you specify a reference head. If the cell water level is above the reference level, then the boundary acts like a normal fixed head boundary conditions. If the head in the cell falls below the reference level, then the boundary condition is turned off. That is, if the simulated head drops below the head reference level the flux is set to zero. Thus, the fixed head drain allows only water extraction.

The reference head can be a prescribed value, fixed at the initial value from the initial conditions, assigned to a .dfs0 time series file, or assigned to a time varying dfs2 file. The last option is typically from a results file. It could be from a regional results file, which can be extracted using the MIKE Zero Toolbox Extraction tool: 2D Grid from 3D files. Or, it could be from a previous run of the same model.

Note: This boundary condition was previously called the Head controlled abstraction boundary condition in early versions of MIKE SHE.

Head Controlled Flux (GHB) - The head controlled flux, or General Internal Head Boundary is similar to the fixed head. However, a flow resistance is incorporated via a user specified leakage coefficient.

The head can be a prescribed value, assigned to a .dfs0 time series file, or assigned to a time varying dfs2 file. The last option is typically from a results file. It could be from a regional results file, which can be extracted using the MIKE Zero Toolbox Extraction tool: 2D Grid from 3D files. Or, it could be from a previous run of the same model.

If the GHB is selected, an extra item is added to the data tree below the boundary condition for the leakage coefficient. The leakage coefficient can be specified as either a simple leakage coefficient [$1/\text{time}$] or as a total conductance [$\text{length}^2/\text{time}$].

Note: This boundary condition was previously called the General Internal Head Boundary condition in early versions of MIKE SHE

Inactive Cells - This boundary condition is used to make interior cells of the model inactive. It works by assigning a hydraulic conductivity of zero to the cells if the simulation is transient, or a value of 10^{-15} if the simulation is steady-state. Note, though, that this method means that in



the pre-processed data, you will see the inactive cells show up in the maps of hydraulic conductivity rather than in the maps of boundary conditions. Also note that since the inactive cells are actually active cells with zero conductivity, the results will also include head values in these points.

Sub-surface drainage and MODFLOW Rivers These two boundary conditions are not yet available.

Related Items:

- Saturated Flow - Reference (V.2 p. 289)
- Boundary Conditions (V.2 p. 296)

2.16.21 Initial concentration

Initial Concentration	
dialogue Type	Stationary Real Data
EUM Data Units	Concentration

There is an initial concentration item for each computational layer. Under each initial concentration item, there is one sub-item for each active species.

The initial concentration is used by the MIKE SHE Water AD engine as the starting concentration for the Water quality simulation.

Related Items:

- Solute Transport in the Saturated Zone (V.2 p. 329)
- Initial Conditions (V.2 p. 338)

2.16.22 Drainage



Saturated zone drainage is a special boundary condition in MIKE SHE used to defined natural and artificial drainage systems that cannot be defined in the MIKE 11 River setup. It can also be used to simulate overland flow in a simple lumped conceptual approach. Surface drainage can only be applied to the top layer of the Saturated Zone model. Water that is removed from the saturated zone by surface drainage is routed to local surface water bodies.

Drainage flow is simulated using an empirical formula, which requires, for each cell, a drainage level and a time constant (leakage factor) that are used for routing the water out of the element. Both drain levels and time constants can be spatially defined. A typical drainage level is 1m below the ground surface and a typical time constant is between $1e-6$ and $1e-7$ 1/s.

MIKE SHE also requires a reference system for linking the drainage to a recipient cell, which can be a MIKE 11 river node, another SZ grid cell, or a model boundary. Whenever drain flow is produced during a simulation, the computed drain flow is routed to the recipient point using a linear reservoir routing technique.

There are three different options for setting up the reference system for drainage

Drainage routed downhill based on adjacent drain levels This option was originally the only option in MIKE SHE. The reference system is created automatically using the slope of the drains calculated from the drainage levels in each cell. Thus, as long as a downward slope is found, the drain flow will continue until crossing a river or the model boundary.

If local depressions in the drainage levels exist, the SZ nodes in these depressions may become the recipients for a number of drain flow producing nodes. This often results in the creation of a lake at such local depressions.

This option is not allowed if using Time varying drainage parameters (*V.I p. 154*).

Drainage routing based on grid codes This method is often used when the topography is very flat, which can result in artificial depressions, or when the drainage system is very well defined, such as in agricultural applications.



In this method, the drainage levels and the time constants are defined as in the previous method. However, a grid code map is also required, which is used to link the drain flow producing cells to a recipient node. The drain levels are still used to calculate the amount of drain flow produced in each node, but the routing is based only on the code values in the drain code file.

Distributed drainage options Choosing this method, adds the Option Distribution item to the data tree. With the Option Distribution, you can specify an integer grid code distribution that can be used to specify different drainage options in different areas of your model.

- If the grid code equals 1, then the first option above is used.
- If the grid code equals 2, then the second method above is used.
- If the grid code equals 3, then the drainage can be routed directly to a particular MIKE 11 branch.
- If the grid code equals 4, then the drainage can be routed to a particular MOUSE manhole.

Drain not routed, but removed from model The fourth option simply exports drainage water out of the model.

Related Items:

- Time varying drainage parameters (*V.1 p. 154*)
- Saturated Zone Drainage (*V.2 p. 298*)
- Drainage with the SOR Solver (*V.2 p. 306*)
- SZ Drainage to Specified MIKE 11 H-points (*V.1 p. 151*)
- Using MIKE SHE with MOUSE (*V.1 p. 181*)

2.16.23 Drain Level

Drain Level	
dialogue Type	Stationary Real Data
EUM Data Units	Elevation

If surface drainage is routed by drain levels, the drainage routing reference system is created automatically using the slope of the drains calculated from the drainage levels in each cell (see Drain flow determined by level of drain). This option was originally the only option in MIKE SHE. The ref-



erence system is created automatically using the slope of the drains calculated from the drainage levels in each cell. Thus, as long as a downward slope is found, the drain flow will continue until crossing a river or the model boundary.).

If surface drainage is routed by grid codes, the drain levels are used to calculate the amount of drain flow produced in each node, but the routing is based only on the code values in the drain code file.

The drain level also determines from which layer the drain water will be extracted.

Related Items:

- Time varying drainage parameters (*V.1 p. 154*)
- Saturated Zone Drainage (*V.2 p. 298*)
- Drainage with the SOR Solver (*V.2 p. 306*)
- SZ Drainage to Specified MIKE 11 H-points (*V.1 p. 151*)
- Using MIKE SHE with MOUSE (*V.1 p. 181*)

2.16.24 Drain Time Constant

Drain Time Constant	
dialogue Type	Stationary Real Data
EUM Data Units	Leakage Coefficient./Drain Time Constant

Drainage flow is simulated using an empirical formula, which requires, for each cell, a drainage level and a time constant (leakage factor). Mathematically, the time constant is exactly the same as a leakage coefficient - it is simply a factor that is used to regulate how quickly the water can drain. A typical time constant is between $1e-6$ and $1e-7$ 1/s.

Related Items:

- Time varying drainage parameters (*V.1 p. 154*)
- Saturated Zone Drainage (*V.2 p. 298*)
- Drainage with the SOR Solver (*V.2 p. 306*)
- SZ Drainage to Specified MIKE 11 H-points (*V.1 p. 151*)
- Using MIKE SHE with MOUSE (*V.1 p. 181*)



2.16.25 Drain Codes

Drain Codes	
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code

If the drainage routing is specified by Drain Codes, a grid code map is required that is used to link the drain flow producing cells to recipient grid cells. The drain levels are still used to calculate the amount of drain flow produced in each node, but the routing is based only on the code values in the drain code file.

The Drain Code can be any integer value, but the different values have the following special meanings:

Code = 0 - Grid cells with an Drain Code value of zero will not produce any drain flow and will not receive any drain flow.

Code > 0 - Grid cells with **positive** Drain Code values will drain to the nearest river, boundary or local depression in the drain level - in that priority - located next to a cell with the same Drain Code value. Thus, if a grid cell produces drainage,

1. If there are one or more cells with the same drain code next to a river link, then the drain flow will be routed to the nearest of these cells.
2. If there are no cells with the same Drain Code located next to a river link, then the drain flow will be routed to the nearest boundary cell with the same Drain Code value.
3. If there are no boundary cells with the same Drain Code value, the drain flow will be routed to the cell with the lowest drain level that has the same Drain Code value (which may create a lake).

Code < 0 - Grid cells with **negative** Drain Code values will drain to either a boundary or a local depression, in that order. Thus, if a grid cell produces drainage,

1. If there are no cells with the same Drain Code located next to a river link, then the drain flow will be routed to the nearest boundary cell with the same Drain Code value.



2. If there are no boundary cells with the same Drain Code value, the drain flow will be routed to the cell with the lowest drain level that has the same Drain Code value (which may create a lake).

One method that is often used is to specify only one Drain Code for the entire model area (e.g. Drain Code 1). Thus, all grids can drain and any drain flow is routed to the nearest river link. If there are no rivers, the drain flow will be routed to the nearest boundary. If you want to route all drain flow to the boundaries instead of the rivers, a negative drain code can be specified for the entire area (e.g. Drain Code -1).

Related Items:

- Saturated Zone Drainage (*V.2 p. 298*)
- Drainage with the SOR Solver (*V.2 p. 306*)
- SZ Drainage to Specified MIKE 11 H-points (*V.1 p. 151*)
- Using MIKE SHE with MOUSE (*V.1 p. 181*)

2.16.26 Option Distribution

Option Distribution	
Conditions:	always when Surface Drainage active AND when the Distributed drainage option is used.
dialogue Type:	Integer Grid Codes
EUM Data Units	Grid Code
Valid Values:	1, 2, 3 and 4 only

The drain type distribution is used to distinguish areas of the model where different drainage options are used.

Code = 1 - Drainage in grid cells with a value of 1 is routed downhill based on the value of the drain level specified in Drain Level data item.

Code =2 - Drainage in grid cells with a value of 2 is routed via Drain Codes as specified in the Drain Codes data item.

Code = 3 - Drainage in grid cells with a value of 3 is routed to a specified MIKE 11 branch and chainage. At the moment, this options requires the use of Extra Parameters.

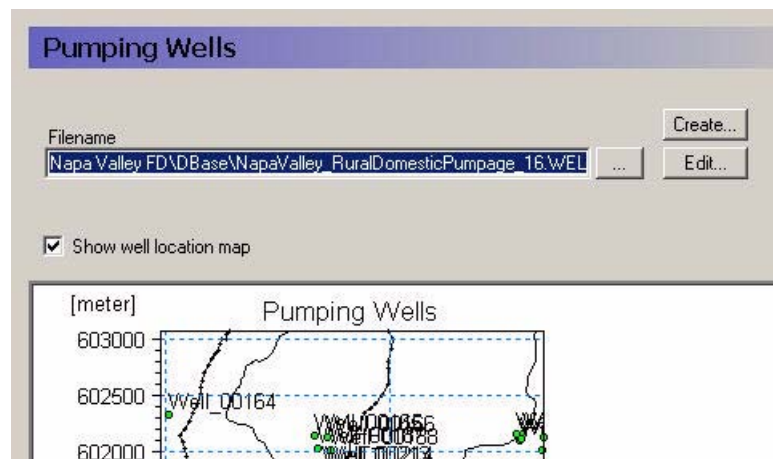


Code = 4 - Drainage in grid cells with a value of 4 is routed to a specified MOUSE man hole. At the moment, this options requires the use of Extra Parameters.

Related Items:

- Saturated Zone Drainage (V.2 p. 298)
- Drainage with the SOR Solver (V.2 p. 306)
- **Extra Parameters (V.2 p. 144)**
- SZ Drainage to Specified MIKE 11 H-points (V.1 p. 151)
- Using MIKE SHE with MOUSE (V.1 p. 181)

2.16.27 Pumping Wells



If pumping wells are active in the model domain, then you must specify the name of the well database to include in the model setup.

Edit The Edit button will open the current Well Database with the current maps and overlays open in the well editor.

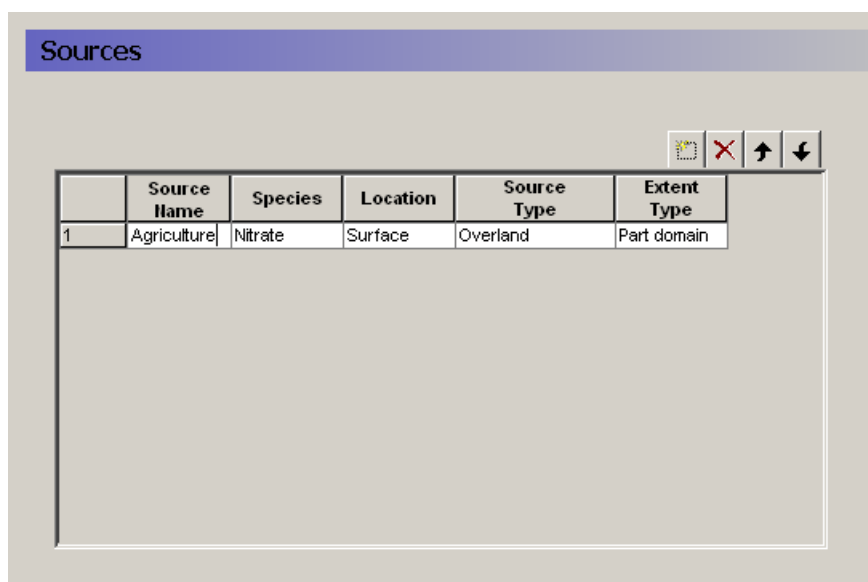
You may get an error in when you open the Well Database if the model has not yet been pre-processed or if the model data has changed without re-preprocessing the model. This error happens because MIKE SHE is trying to reconcile and plot both the geologic layers (input data) and the numerical layers (preprocessed data) for each cell containing a pumping well.

Create The Create button will create a new Well Database file

**Related Items:**

- Well editor (V.2 p. 167)
- 3D Finite Difference Method (V.2 p. 289)
- Boundary Conditions (V.2 p. 296)
- Linear Reservoir Method (V.2 p. 307)
- Calculation of Baseflow (V.2 p. 316)

2.17 Sources



The specification of water quality sources is very flexible. The Sources dialogue allows you to add and delete sources, as well as define the type and location of the source. The table provides an overview of all of your sources in your model.

An important feature of the source location definition is the partial extent distribution function. This allows you to define, for example, a distributed global source file - say of the field scale agricultural inputs in your catchment - and run individual water quality scenarios for each sub-catchment (modelled as an partial extent) to assess the subcatchment contributions to the global stream impact.

Source Name - The name appears in the data tree for reference.



Species - You can only choose from the list of available species that you have defined in the **Species (V.2 p. 50)** dialogue.

Location - The location is defines whether the source is located on the ground surface (Surface) or in the saturated or unsaturated zone (Sub-surface). The available source types depend on where the source is located.

Source Type - If the source is located on the ground surface, then it can be either a Precipitation source (concentration in precipitation water) or an Overland source (mass on the surface). In both cases, the solute can infiltration or runoff as lateral overland flow.

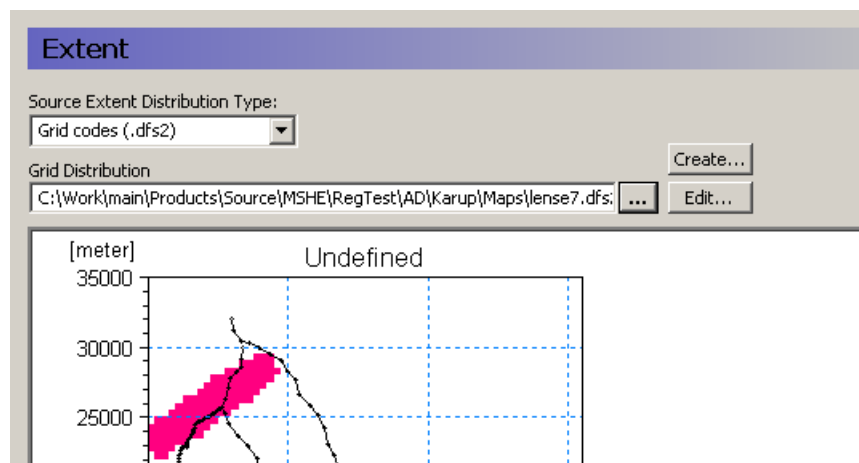
Extent Type - The source can cover the entire domain (Full domain) or only part of the domain (Part domain). In both cases, the actual source strength can vary spatially and temporally. The Extent is used simply to restrict the source data to a zone smaller than the model domain.

Related Items:

- Solute Transport in the Saturated Zone (*V.2 p. 329*)
- Solute Transport in the Unsaturated Zone (*V.2 p. 340*)
- Solute Transport in Overland Flow (*V.2 p. 344*)



2.17.1 Extent



Extent

Conditions:	If source extent is Part Domain
dialogue Type	Integer Grid Code
EUM Data Units	Integer Grid Code

If the source extent is Part Domain, then you can define a local extent for which the larger source file is applied. This allows you to define, for example, a distributed global source file - say of the field scale agricultural inputs in your catchment - and run individual water quality scenarios for each sub-catchment (modelled as an partial extent) to assess the subcatchment contributions to the global stream impact.

Related Items:

- Solute Transport in the Saturated Zone (*V.2 p. 329*)
- Solute Transport in the Unsaturated Zone (*V.2 p. 340*)
- Solute Transport in Overland Flow (*V.2 p. 344*)



2.17.2 Top Elevation

Top Elevation	
Conditions:	If source location is Subsurface.
dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Height above ground

The Top Elevation refers to the upper elevation of a solute source. It is used by the interpolation algorithm to assign a source location to the saturated or unsaturated zone cells.

Related Items:

- Solute Transport in the Saturated Zone (*V.2 p. 329*)
- Source/Sinks, Boundary Conditions and other Exchanges (*V.2 p. 338*) in the saturated zone
- Solute Transport in the Unsaturated Zone (*V.2 p. 340*)
- Source/Sinks, Boundary Conditions and other Exchanges (*V.2 p. 343*) in the unsaturated zone

2.17.3 Bottom Elevation

Bottom Elevation	
Conditions:	If source location is Subsurface.
dialogue Type	Stationary Real Data
EUM Data Units	Elevation or Height above ground

The Bottom Elevation refers to the lower elevation of a solute source. It is used by the interpolation algorithm to assign a source location to the saturated or unsaturated zone cells.

Related Items:

- Solute Transport in the Saturated Zone (*V.2 p. 329*)
- Source/Sinks, Boundary Conditions and other Exchanges (*V.2 p. 338*) in the saturated zone
- Solute Transport in the Unsaturated Zone (*V.2 p. 340*)



- Source/Sinks, Boundary Conditions and other Exchanges (*V.2 p. 343*) in the unsaturated zone

2.17.4 Strength

Strength	
dialogue Type:	Time-varying Real Data
EUM Data Units	Grid Code
Time Series EUM Data Units	Concentration

Since the source is a common data item for overland, unsaturated and saturated flow, the units of the source strength depend on the type of source being simulated.

The source strength comprises both a distribution and a value. The distribution can be either uniform, station-based or fully distributed. If the data is station-based then for each station a sub-item will appear where you can enter the time series of values for the station. If the data is fully distributed, then you can enter a time varying dfs2 file.

Related Items:

- Solute Transport in the Saturated Zone (*V.2 p. 329*)
- Source/Sinks, Boundary Conditions and other Exchanges (*V.2 p. 338*) in the saturated zone
- Solute Transport in the Unsaturated Zone (*V.2 p. 340*)
- Source/Sinks, Boundary Conditions and other Exchanges (*V.2 p. 343*) in the unsaturated zone
- Solute Transport in Overland Flow (*V.2 p. 344*)
- Source/Sinks, Boundary Conditions and other Exchanges (*V.2 p. 348*) in overland flow



2.18 Storing of Results

Default output folder - If you unselect this option, then you can change the output folder for the results. If you change the output folder, then you must re-run the model for the Results tab to point to the correct folder.

Water Movement Output

Storing of water balance data - When this option is selected, MIKE SHE will store all of the relevant output data for the analysis of the water balance. This will automatically select the required items in the Grid Series Output section.

Storing of Hot start data - The option allows you to save a simulation that can be used as the start a new simulation. See Time Step Control for more information on using Hot Start data as the initial data for a simulation.

Only store hot start data at the end of the simulation - Typically, the follow on simulation starts at the end of the previous simulation. However, if you want to test the sensitivity of the results to the starting condition, for example, you may want to save hot start data more



frequently. However, the hot start file can become very large if the hot start data is saved frequently.

Store AD input data during water movement simulation - A MIKE

SHE water quality simulation is calculated based on the cell-by-cell water fluxes calculated by the water movement module. If water quality is included in the model setup, then the necessary data is automatically saved and this item is hidden. However, if water quality is not included in the simulation, you can optionally tell MIKE SHE to save the necessary data for the water quality model by checking this box on and selecting the save option. The first option only saves the saturated zone data, which is suitable if you are only going to calculate the water quality in the saturated zone. For example, the random walk particle tracking is only available in the saturated zone and there is, therefore, no need to save overland flow data for a particle tracking simulation. The second option saves all of the data necessary for a water quality simulation in the complete integrated model.

Storing interval for grid series output

Gridded output can create very large output files if the data is stored at every time step. Gridded data is not usually compared to frequent measurements, such as daily groundwater levels, so the output frequency can be much less than the time step length. In fact, the output frequency of gridded output is often determined by visualization needs, such as to make smooth animations.

The gridded output for the different processes can be saved at different frequencies - the overland frequency is separate from unsaturated zone frequency is separated from the saturated zone frequency. However, you cannot save individual items at different frequencies. Thus, since precipitation, evapotranspiration and unsaturated flow output items are related, they are all saved at the same frequency. The saturated zone heads and fluxes are separated into different frequencies, however, because the gridded output files for a detailed 3D model can get very large.

Note: The Storing Time Step for SZ must be an integer multiple of the Maximum Allowed SZ Time Step that is specified in the Time Step Control dialogue. In other words, if the Maximum allowed SZ time step is 24 hrs, then the Storing Time Step for SZ can only be a multiple of 24 hours (e.g. 24, 48, 72 hours, etc.).

Similarly, the storing frequency for SZ Fluxes must be an integer multiple of the SZ heads frequency, and the storing frequency for overland flow and unsaturated flow items, must be an integer multiple of their respective maximum allowed time step length.



Water Quality Output

Water Quality output

Storing interval for grid series output

Overland (OL): 24 (hrs) Unsaturated Zone: 24 (hrs) Saturated Zone: 24 (hrs)

Storing interval for mass balance output

Time Series(.dfs0): 24 (hrs) Summary(ASCII): 24 (hrs)

Storing interval for grid series output - similar to the storing interval for water movement data, the gridded output data files for a detailed transport simulation can get very large. These three frequencies allow you to save only the data you need.

Storing interval for mass balance output - similar to the storing interval for grid series output, these separate storing frequencies allow you to save only the data you need.

Related Items:

- Time Step Control (*V.2 p. 30*)
- Detailed time series output (*V.2 p. 138*)
- Grid series output (*V.2 p. 143*)



2.18.1 Detailed time series output

	Name	Data type	New plot	X	Y	Depth	
1	146.532 DK5	head elevation in saturated zone	<input checked="" type="checkbox"/>	590623	6.13409e+0	20	
2	146.1936 DK	head elevation in saturated zone	<input checked="" type="checkbox"/>	594593	6.13035e+0	35	
3	146.2040 DK	head elevation in saturated zone	<input checked="" type="checkbox"/>	588873	6.1288e+00	54	
4	146.633 DK3	head elevation in saturated zone	<input checked="" type="checkbox"/>	589597	6.12972e+0	21	
5	146.679 DK2	head elevation in saturated zone	<input checked="" type="checkbox"/>	590577	6.13047e+0	27	

	X	Y	Depth		Incl. Obs. Data	Obs. Data Filename			
1	590623	6.13409e+0	20		<input checked="" type="checkbox"/>	C:\5.Testing\...\Kalibreringspej	...	Edit...	New...
2	594593	6.13035e+0	35		<input checked="" type="checkbox"/>	C:\5.Testing\...\Kalibreringspej	...	Edit...	New...
3	588873	6.1288e+00	54		<input checked="" type="checkbox"/>	C:\5.Testing\...\Kalibreringspej	...	Edit...	New...
4	589597	6.12972e+0	21		<input checked="" type="checkbox"/>	C:\5.Testing\...\Kalibreringspej	...	Edit...	New...
5	590577	6.13047e+0	27		<input checked="" type="checkbox"/>	C:\5.Testing\...\Kalibreringspej	...	Edit...	New...

The Detailed time series output dialogue allows you to specify the location at which you want detailed time series output and the item that you want output. For each specified point, the output variable is stored in a .dfs0 file with one value for every simulation time step. Finally, for each item in the detailed time series table, an HTML plot is created in the Result Tab.

Note All of the detailed time series items are stored in one .dfs0 file. This can lead to file size and disk space errors, if you have a long detailed simulation or more than ~200 detailed time series items. Also, the HTML output in the Results Tab will become very slow if you have a lot of items, since it has to read the entire .dfs0 file and generate all of the graphs every time you access the Detailed Time Series page in the Results Tab.

Name - This is a text field that can be used to specify a reference name for the location, for example, a borehole name. This is also the name that will be used for the time series item in the Dfs0 file created during the simulation.

Data Type - This is the list of available output items is dynamic in the sense that the list changes in response to the processes included in the Simulation Specification dialogue. Further, additional items are available that are related to simulation variables, such as the number of iterations during each Saturated Zone time step. A list of available Data Types can be found in Output Items (V.1 p. 87).

New plot - If this is checked, then a new detailed time series HTML-plot will be created on the Results Tab. If this is unchecked, then the



output will be added to the previous plot. You can use the Up and Down arrows to arrange the output points so that relevant points are plotted together.

X, Y - Often, detailed time series are associated with measurement stations. That is, locations at which a time series of measurements are available, for example, water levels in a well or water depths on a flood plain. This is the (X, Y) map coordinates of the point in the same EUM units (ft, m, etc.) as specified in the EUM Database for Item geometry 2-dimensional. (see EUM Data Units)

Depth - This is the depth of the observation point below land surface for subsurface observation points. The value is in same EUM units (ft, m, etc.) as specified in the EUM Database for Depth Below Ground (see EUM Data Units).

Target Icon - You can use the target icon to locate the output point exactly. Alternatively, you can type the exact coordinates or import the items from an ASCII file.

Include observation data - If this is checked, then a .dfs0 file can be specified that includes observation points. The observation points are automatically plotted along with the results in the HTML plot on the Results tab. The .dfs0 item is selected in the file browser dialogue. The **Edit** button opens the specified .dfs0 file and the **New** button can be used to create a new .dfs0 file with the correct item type etc. and at the same time import data from an Excel spreadsheet.

Importing data

Detailed MIKE SHE Time Series data can be imported directly into the Detailed MIKE SHE Time Series dialogue using the Import button. The data file must be a tab-delimited ASCII file without a header line. The file must contain the following fields and be in the format specified below.

```
Name>DataTypeCode>NewPlot>X>Y>Depth>UseObsdata>dfs0file name>dfs0ItemNumber
```

where the > symbol denotes the Tab character and

Name - is the user specified name of the observation point. This is the name that will be used for the time series item in the Dfs0 file created during the simulation.

DataTypeCode - This is a numeric code used to identify the output data type. A list of available Data Type Codes can be found in Output Items (V.1 p. 87).



NewPlot - This is a flag to specify whether a new detailed time series HTML-plot will be created on the Results Tab:

0 = the output will be added to the previous plot.

1 = Create a new plot

X, Y - This is the (X, Y) map coordinates of the point in the same EUM units (ft, m, etc.) as specified in the EUM Database for Item geometry 2-dimensional. (see EUM Data Units)

Depth - This is the depth of the observation point below land surface for subsurface observation points. The value is in same EUM units (ft, m, etc.) as specified in the EUM Database for Depth Below Ground (see EUM Data Units). A depth value must always be included.

UseObsData - This is a flag to specify whether or not an observation file needs to be input: 0 = No; 1 = Yes

dfs0file name - This is the file name of the dfs0 time series file with observation data. The path to the dfs0 file must be relative to the directory containing the MIKE SHE *.she document. The .dfs0 extension is added to the file name automatically and should not be included in the file name. For example,

.\Time\Calibration\GroundwaterObs

refers to the file *GroundwaterObs.dfs0* located in the subdirectory *Time\Calibration*, which is found in the same directory as the .she model document.

dfs0ItemNumber - This is the Item **number** of the observation data in the specified DFS0 file.

The following is a simple example with three MIKE SHE observation points, where the file name is obsdata.dfs0

```
Obs_1 >20 >1 >234500. >456740. >0. >0 >.\time\obsdata >1
Obs_2 >15 >1 >239700. >458900. >10. >1 >.\time\obsdata >2
Obs_3 >16 >0 >241500. >459310. >20. >1 >.\time\obsdata >3
```

Related Items:

- Time Step Control (V.2 p. 30)
- Detailed MIKE 11 Output (V.2 p. 141)



- Grid series output (V.2 p. 143)
- MIKE SHE Detailed Time Series (V.2 p. 156)

2.18.2 Detailed MIKE 11 Output

Detailed M11 timeseries output									
Import...									
	Name	Data type	Branch name	Chainage	Incl. Obs. Data	Obs. Data Filename			
1	Rhein at Koln	Water Level	Rhein - main	134560	<input checked="" type="checkbox"/>	C:\5.Testing\TS1.dfs0	...	Edit...	New...
2	Rhein at Bonn	Discharge	Rhein - main	156740	<input checked="" type="checkbox"/>	C:\5.Testing\TS1.dfs0	...	Edit...	New...

The Detailed time series output for MIKE 11 allows you to specify the river chainage location at which you want detailed time series output and the item that you want output. For each specified point, the output variable is stored in a .dfs0 file with one value for every simulation time step. Finally, for each item in the detailed time series table, an HTML plot is created in the Result Tab - with or without observation data.

The principle advantage of this option is that you can now easily create calibration plots of calculated versus observed water levels without opening and having to create specific plots in MIKE View.

Name - This is a text field that can be used to specify a reference name for the location, for example, a gage name.

Data Type - This is the list of available output items, which for MIKE 11 only contains two items - water level and flow rate.

Branch name - the Branch name must be a valid branch name in the MIKE 11 model. However, this is not checked until run time, at which point an error message will be generated if it is not valid and the simulation will be stopped.

Chainage - like the branch name, the chainage must be a valid MIKE 11 chainage.

Include observation data - If this is checked, then a .dfs0 file can be specified that includes observation points. The observation points are automatically plotted along with the results in the HTML plot on the Results tab. The .dfs0 item is selected in the file browser dialogue. The **Edit** button opens the specified .dfs0 file and the **New** button can be



used to create a new .dfs0 file with the correct item type etc. and at the same time import data from an Excel spreadsheet.

Importing data

Detailed MIKE 11 Time Series data can be imported directly into the Detailed MIKE 11 Time Series dialogue using the Import button. The data file must be a tab- delimited ASCII file without a header line. The file must contain the following fields and be in the format specified below.

Name>DataTypeCode>BranchName>Chainage>UseObsdata>dfs0file name>dfs0ItemNumber

where the > symbol denotes the Tab character and

Name - is the user specified name of the observation point. This is the name that will be used for the time series item in the Dfs0 file created during the simulation.

DataTypeCode - This is a numeric code used to identify the output data type.

Branch name - the Branch name must be a valid branch name in the MIKE 11 model. However, this is not checked until run time, at which point an error message will be generated if it is not valid and the simulation will be stopped.

Chainage - like the branch name, the chainage must be a valid MIKE 11 chainage.

UseObsData - This is a flag to specify whether or not an observation file needs to be input: 0 = No; 1 = Yes

dfs0file name - This is the file name of the dfs0 time series file with observation data. The path to the dfs0 file must be relative to the directory containing the MIKE SHE *.she document. The .dfs0 extension is added to the file name automatically and should not be included in the file name. For example,

.\Time\Calibration\RiverstageObs

refers to the file *RiverStageObs.dfs0* located in the subdirectory *Time\Calibration*, which is found in the same directory as the .she model document.

dfs0ItemNumber - This is the Item **number** of the observation data in the specified DFS0 file.



The following is a simple example with three MIKE 11 observation points, where the file name is obsdata.dfs0

```
Obs_1 > ? > GrandRiver > 34500. >0 >.\time\obsdata >1
Obs_2 > ? > GrandRiver > 22500. >1 >.\time\obsdata >2
Obs_3 > ? > GrandRiver > 1500. >1 >.\time\obsdata >3
```

Related Items:

- Time Step Control (*V.2 p. 30*)
- Detailed time series output (*V.2 p. 138*)
- Grid series output (*V.2 p. 143*)
- MIKE 11 Detailed Time Series (*V.2 p. 159*)

2.18.3 Grid series output

	Enable	Item	Required for	
5	<input type="checkbox"/>	crop coefficient		Na
6	<input type="checkbox"/>	actual evapotranspiration		Na
7	<input checked="" type="checkbox"/>	actual transpiration	Water Balance	Na
8	<input checked="" type="checkbox"/>	actual evaporation from interception	Water Balance	Na
9	<input checked="" type="checkbox"/>	actual evaporation from ponded water	Water Balance	Na
10	<input checked="" type="checkbox"/>	canopy interception storage	Water Balance	Na
11	<input checked="" type="checkbox"/>	evapotranspiration from SZ	Water Balance	Na
12	<input checked="" type="checkbox"/>	depth of overland water	Water Balance	Na
13	<input checked="" type="checkbox"/>	overland flow in x-direction	Water Balance	Na
14	<input checked="" type="checkbox"/>	overland flow in y-direction	Water Balance	Na
15	<input checked="" type="checkbox"/>	External sources to Overland (for OpenMI)	Water Balance	Na
16	<input checked="" type="checkbox"/>	Water content in root zone (2-layer UZ)		Na
17	<input type="checkbox"/>	Water content below root zone (2-layer UZ)		Na
18	<input type="checkbox"/>	Maximum water content (2-layer UZ)		Na
19	<input type="checkbox"/>	Minimum water content (2-layer UZ)		Na

The Grid series output dialogue allows you to specify the frequency at which you want detailed output of gridded data and the items that you want output. A list of available Data Types can be found in Output Items (*V.1 p. 87*). The list is dynamic in the sense that the list changes in response to the processes included in the Simulation Specification dialogue.

In some cases, such as when the Water Balance output has been specified (see Storing of Results), some of the items will be automatically selected and cannot be unselected. This will be noted in the Required for column of the dialogue.

**Related Items:**

- Time Step Control (*V.2 p. 30*)
- Detailed time series output (*V.2 p. 138*)
- Detailed MIKE 11 Output (*V.2 p. 141*)
- Gridded Data Results Viewer (*V.2 p. 157*)

2.19 *Extra Parameters*

The Extra Parameters Section is available to support new features in MIKE SHE that are not yet supported in the dialogues and data tree. Detailed descriptions of the features that use Extra Parameters are found in Extra Parameters (*V.1 p. 145*).

If you need to activate a feature that is only supported in the Extra Parameters section, you must first add the necessary number of lines to the Extra Parameters table. Then fill in the data that is required for the module.

Name - this is the name of the parameter that is required by the unsupported feature. It must be spelled exactly as specified in the documentation. It may be the actual name of the feature or the name of a parameter.

Type - The type is the type of parameter. The following types are available:

Float - Real, floating point number

Integer - Integer number

Boolean - an On/Off checkbox - typically used to turn a feature on or off

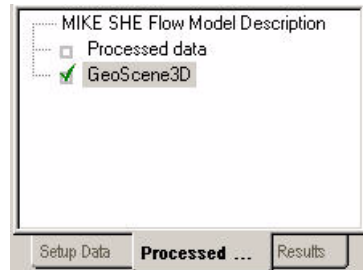
Text - a character string

file name - this is typically the file where more detailed input data is recorded

Value - this is the value associated with the Type above



3 **PREPROCESSED DATA TAB**



In the Setup Tab, you specify the input data required by the model - including the Model Domain and Grid. However, most of the Setup Data is independent of the Model Domain and Grid. When you pre-process your model set up, MIKE SHE's pre-processor program scans through your model set up and interpolates all spatial data to the specified model domain and grid. This interpolated set up data is stored in a .fif file, which is read during the simulation by the MIKE SHE engine. However, the .fif file does not include any time information. All time series information is interpolated dynamically during the run. This is necessary because the time steps in MIKE SHE can dynamically change during the simulation in response to stresses on the system.

The Preprocessed Data Tab is used to display the spatial content of the .fif file.

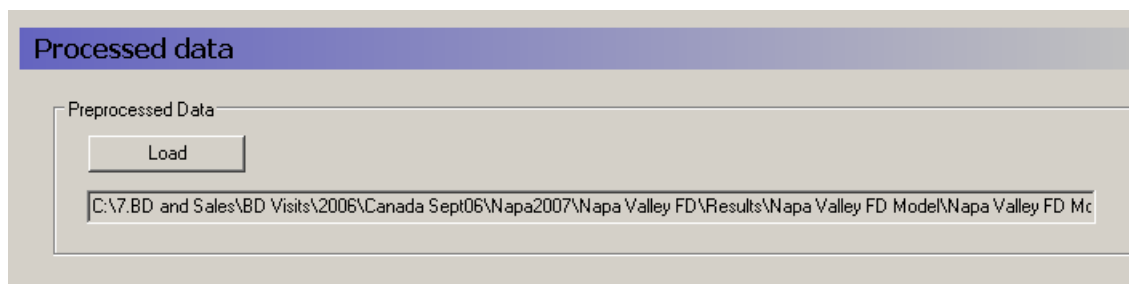
Before you run your simulation, you should carefully check the preprocessed data for errors. Errors found in the preprocessed data are typically related to incorrectly specified parameters, file names, etc. in the Setup Tab.

The Preprocessed Tab includes a data tree with two items:

- Processed Data (*V.2 p. 146*)
- GeoScene3D (*V.2 p. 152*)



3.1 *Processed Data*



There is only one button in the main dialogue for the processed data dialogue, plus an uneditable text box displaying the current .fif file created by the pre-processor.

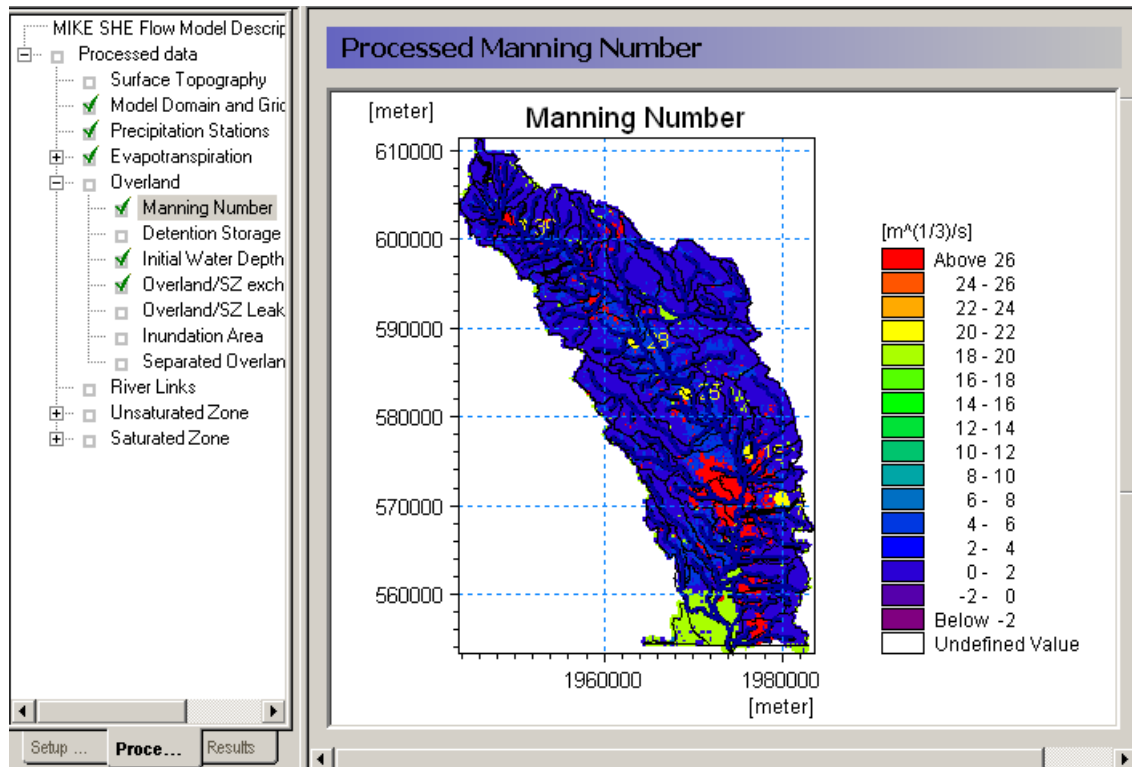
Load - After you have pre-processed your model setup, and a .fif file is created, you can click on the Load button to load the contents of the .fif file and view the actual model input data.

If the model Setup data has been changed since the last pre-processing, you will get a warning message telling you that the pre-processed data may not match the current setup data.

Note If you have changed anything in your model setup, and then run the pre-processor again, you must re-load the new .fif file to be able to see the changes in the preprocessed view.



Pre-processed Data View



Once the pre-preprocessed data in the .fif file has been loaded, then the data tree reflects all the spatial data defined in the model set up tab. In other words, if the overland flow is not included in the **Simulation Specification (V.2 p. 26)** dialogue, then the Overland item will not be included in the pre-processed data tree.

3.1.1 Model Domain and Grid

The model domain and grid item displays the grid code values required for the MIKE SHE model. This differs slightly from the **Model Domain and Grid (V.2 p. 52)** item in the Setup Tab. In the .fif file, all cells outside the model domain are assigned a value of zero, compared to the Setup tab where the cells outside of the model boundary are delete values.

Unlike other data items in the pre-processed tab, you cannot save the pre-processed model domain and grid to a dfs2 file and re-use it in the Setup Tab, because the **Model Domain and Grid (V.2 p. 52)** item requires delete values outside of the model domain.

**Related Items:**

- **Model Domain and Grid (V.2 p. 52)**

3.1.2 Precipitation and Evapotranspiration

The precipitation and evapotranspiration items display the integer station codes for the time series defined in Precipitation Rate (V.2 p. 58) and Reference Evapotranspiration (V.2 p. 79) items in the Setup Tab. The station names are not displayed, so you will have to refer back to the Setup Tab for the station names

However, the .fif file does not include any time information. All time series information is interpolated dynamically during the run. This is necessary because the time steps in MIKE SHE can dynamically change during the simulation in response to stresses on the system.

Related Items:

- **Precipitation (V.2 p. 57)**
- **Evapotranspiration (V.2 p. 79)**

3.1.3 River Links

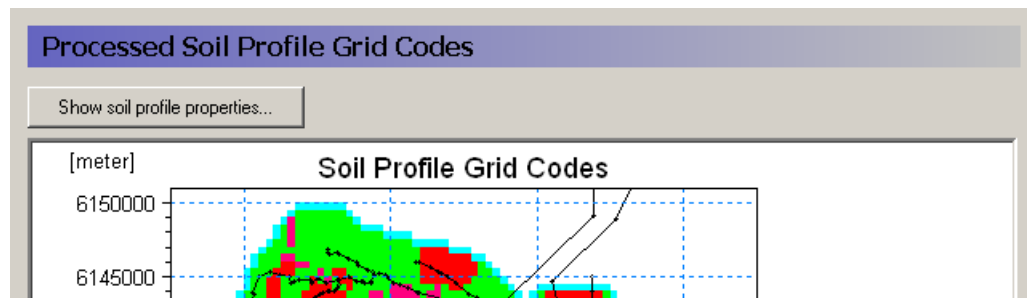
The coupling between MIKE 11 and MIKE SHE is made via river links, which are located on the edges that separate adjacent grid cells. The river link network is created by the pre-processor, based on the MIKE 11 coupling reaches. The entire river system is always included in the hydraulic model, but MIKE SHE will only exchange water with the coupling reaches.

The location of each of MIKE SHE river link is determined from the coordinates of the MIKE 11 river points, where the river points include both digitised points and H-points on the specified coupling reaches. Since the MIKE SHE river links are located on the edges between grid cells, the details of the MIKE 11 river geometry can be only partly included in MIKE SHE, depending on the MIKE SHE grid size. The more refined the MIKE SHE grid, the more accurately the river network can be reproduced. This also leads to the restriction that each MIKE SHE grid cell can only couple to one coupling reach per river link. Thus, if, for example, the distance between coupling reaches is smaller than half a grid cell, you will probably receive an error, as MIKE SHE tries to couple both coupling reaches to the same river link.

The river links are shown on all the maps and the distributed data shown on the River Links map is the Topography.

**Related Items:**

- **Rivers and Lakes** (V.2 p. 80)
- Coupling of MIKE SHE and MIKE 11 (V.2 p. 228)

3.1.4 UZ Soil Profile Grid Codes

The unsaturated zone is composed of 1D soil columns. If you are using the Richards equation or the Gravity flow method, then these columns consist of a vertical grid with various soil properties. The **Show soil profile properties** button located just above the map allows you to view a summary of the unsaturated zone grid for each cell. If you click on this button, the cursor will change to a target icon. When you click on a particular cell, an ASCII txt file will be created and opened, which contains the summary data.

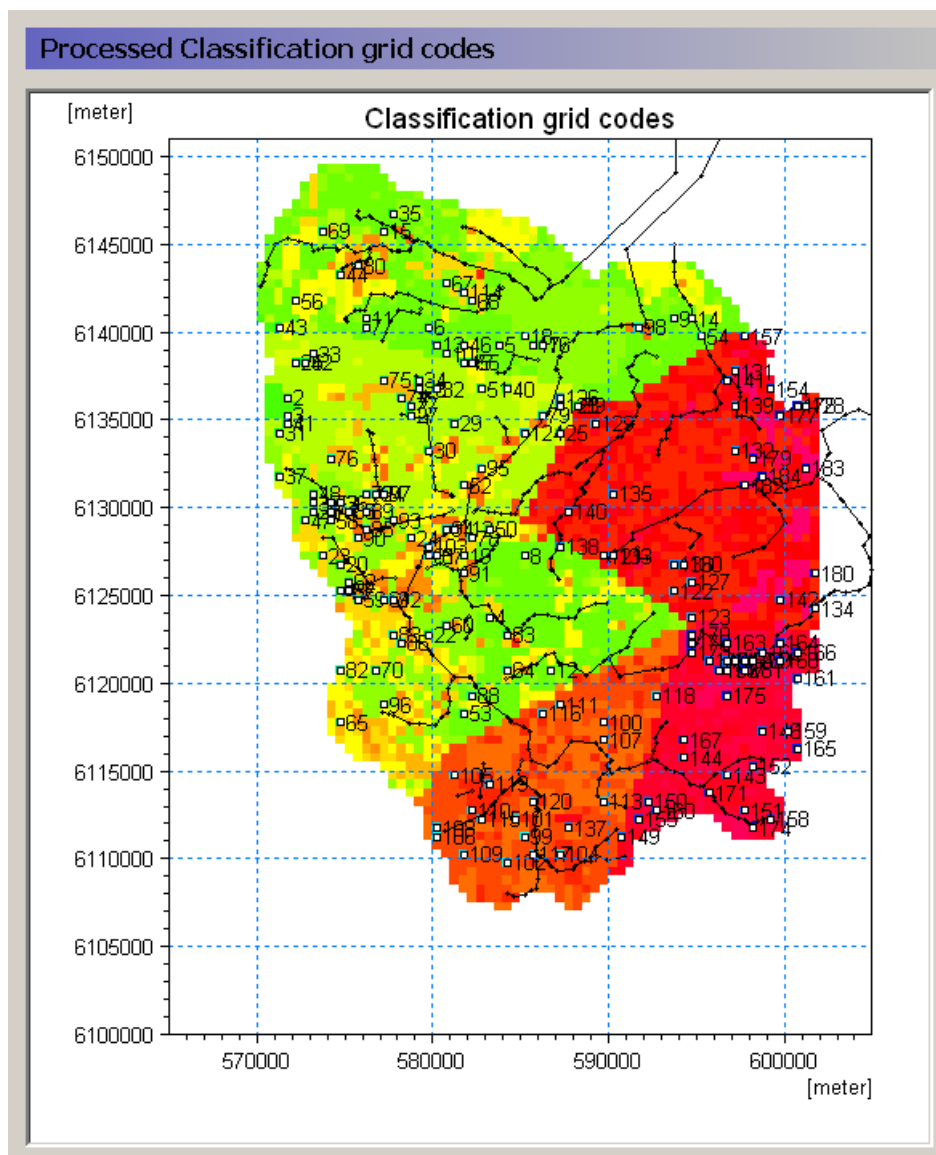
Note that the pre-processor modifies the vertical discretisation wherever the vertical cell size changes. Thus, if you have 10 cells of 20cm thickness, followed by 10 cells of 40cm thickness, the location of the transition will be moved such that the two cells on either side will have an equal thickness. In this case, cells 10 and 11 will both be 15cm.

Related Items:

- Soil Profile Definitions (V.2 p. 93)
- Richards Equation (V.2 p. 262)
- Gravity Flow (V.2 p. 273)



3.1.5 UZ Classification Grid Codes



If certain conditions are met, then the flow results for a 1D unsaturated zone column can be applied to columns with similar properties. In this map, each numbered item is a calculation point. The cell with a calculation point is given an integer grid code with a negative value. The flows calculated during the simulation in the cells with the negative code, will be transferred to all the cells with the same positive grid code value. For example, if an UZ recharge to SZ of $0.5 \text{ m}^3/\text{day}$ is calculated for UZ grid



code -51, then all the SZ cells below the UZ cells with a grid code of +51 will also be given the same recharge.

By just looking at the map it can be difficult to distinguish which calculations are being transferred to which cells. An easier way to look at this is to save the map to a dfs2 file (right click) and then open the file in the Grid Editor, where it is much easier to search for cell numbers.

Related Items:

- Soil Profile Definitions (V.2 p. 93)
- **Unsaturated Zone (V.2 p. 90)**
- Column Classification (V.2 p. 91)
- Partial automatic classification (V.2 p. 97)
- Specified classification (V.2 p. 98)

3.1.6 Saturated Zone Items

The saturated zone items are organized by item with separate maps per layer.

Layer thickness

Layer thickness is a derived value calculated by subtracting the top and bottom elevations of the layer.

The EUM type is “thickness”, with a MIKE Zero default unit of millimeters. This is not very suitable for geologic layers that can be more than 100m thick. To change the default units, see EUM Data Units (V.1 p. 271), find the ‘thickness’ item and change the default unit to meter or feet, as appropriate.

Transmissivity

Transmissivity is also a derived value calculate by multiplying the thickness by the horizontal hydraulic conductivity.

3.1.7 Saturated Zone Drainage

The rate of saturated zone drainage is controlled by the drain elevation and the drain time constant. However, the destination of the drainage water is controlled by drain codes, which determine if the water flows to a boundary, a local depression, or a river.

SZ Drainage Codes

The SZ Drainage Codes map is the drainage codes specified in the Drain Codes (V.2 p. 127) set up item, interpolated to the model grid.



During the preprocessing, each active drain cell is mapped to a destination cell. The destination cell is determined from the drain code values and the proximity of rivers and boundaries. Then, whenever drainage is generated in a cell, the drain water will always be routed to the same destination cell.

Drainage to local depressions and boundary

All cells with the same positive code are drained to the cell with the same numeric negative code.

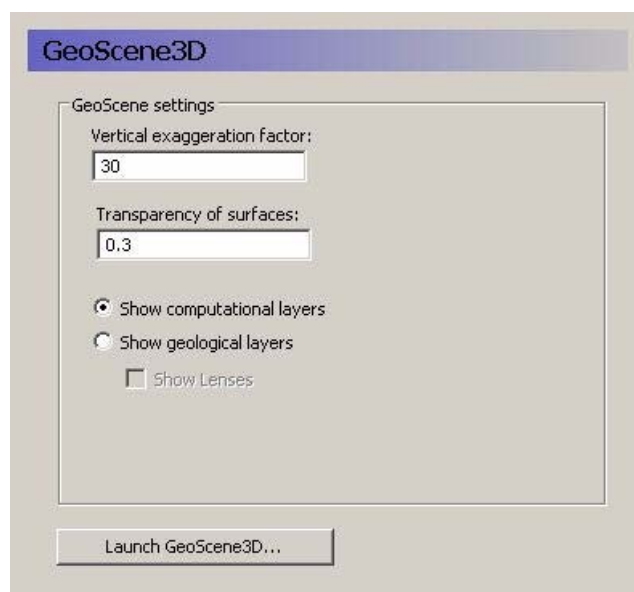
Drainage to river

All cells with the same positive code are drained to the cell with the same numeric negative code.

Related Items:

- Groundwater Drainage (*V.1 p. 53*)
- Drainage (*V.2 p. 123*)
- Drain Level (*V.2 p. 125*)
- Drain Time Constant (*V.2 p. 126*)
- Drain Codes (*V.2 p. 127*)
- Option Distribution (*V.2 p. 128*)

3.2 GeoScene3D





The GeoScene3D dialogue provides a link to the GeoScene3D program for viewing the preprocessed model setup. The preprocessed data does not include any transient or time data. So, GeoScene3D only shows stationary data.

Vertical exaggeration factor - In most models, the lateral extent is several orders of magnitude greater than the vertical extent. Thus, without a vertical exaggeration factor, the model view will be too thin to be useful. A vertical exaggeration of 10 to 30 will typically give you a good looking model that accentuates the vertical differences.

Transparency of surfaces - The transparency factor allows you to see through the model surfaces. This gives you a better feeling of what is happening below or above the surface that you are looking at.

Show computational layers - In this case, the model's computational layers will be visible and selectable in the GeoScene3D interface.

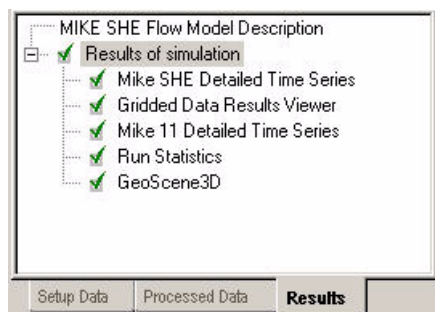
Show geological layers - In this case, the specified conceptual geologic layers will be visible and selected in the GeoScene3D interface.

Show lenses - If you select to show lenses, then they will be displayed on top of the geologic layers.





4 RESULTS TAB



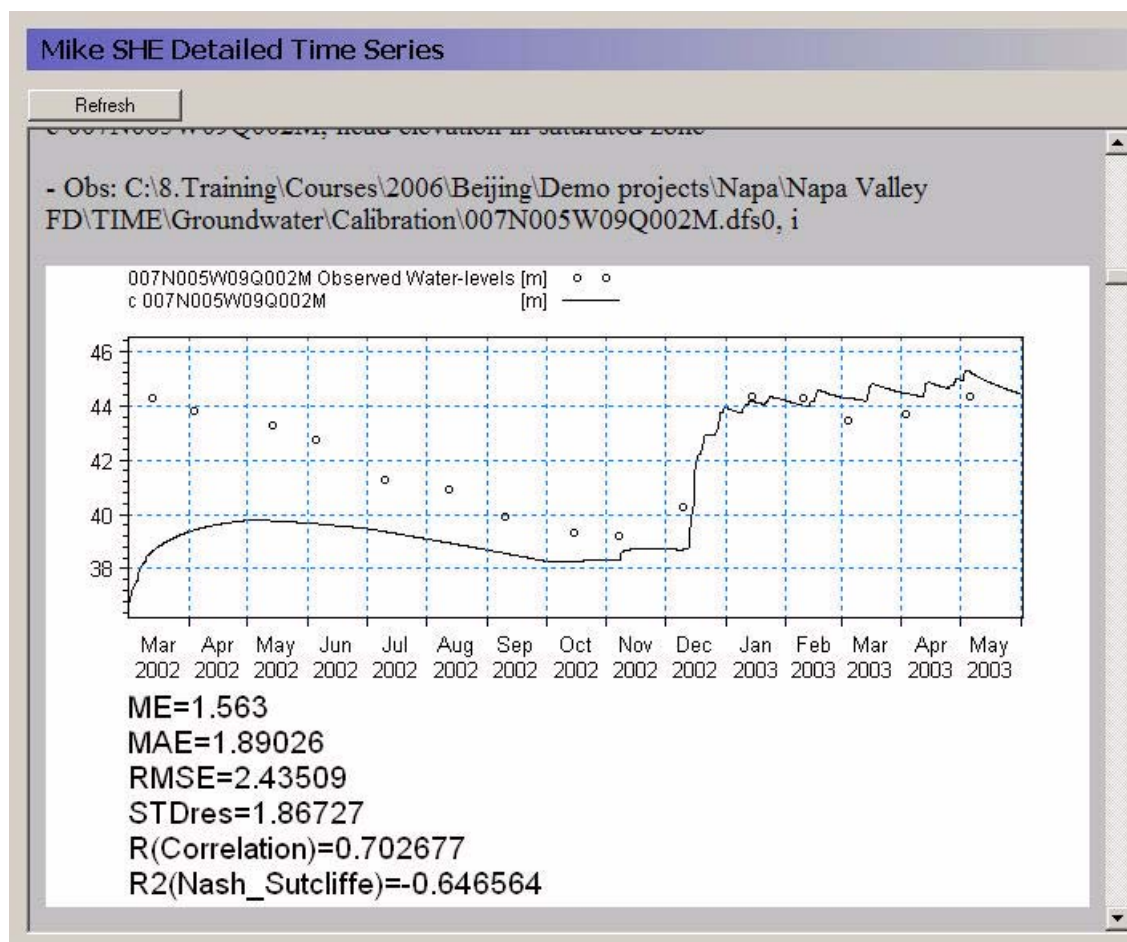
All the simulation results are collected in the Results tab. This includes Detailed time series output for both MIKE SHE and MIKE 11, as well as Grid series output for MIKE SHE.

A Run Statistics tool is available for helping you assimilate the calibration statistics for each of the detailed time series plots.

A link to the GeoScene3D program is also included, where you can visualize your results in a dynamic 3D environment.



4.1 MIKE SHE Detailed Time Series



The MIKE SHE Detailed time series tab includes an HTML plot of each point selected in the Detailed time series output (V.2 p. 138) dialogue. The HTML plots are updated during the simulation whenever you enter the view. Alternatively, you can select the Refresh button to refresh the plot.

Note that the HTML plot is regenerated every time you enter the view. So, if you have a lot of plots and a long simulation, then the regeneration can take a long time.

Related Items:

- Detailed time series output (V.2 p. 138)
- Statistic Calculations (V.2 p. 161)



4.2 Gridded Data Results Viewer

Gridded Data Results Viewer				
Layer no. for Groundwater items				
1				
	Item	Add XY-flow vectors		Filename
1	precipitation rate	<input type="checkbox"/>	View result...	C:\5.Testing\NRSøby\NrSøby2003-7layer-250b\NrSøby200
2	rooting depth	<input type="checkbox"/>	View result...	C:\5.Testing\NRSøby\NrSøby2003-7layer-250b\NrSøby200
3	leaf area index	<input type="checkbox"/>	View result...	C:\5.Testing\NRSøby\NrSøby2003-7layer-250b\NrSøby200
4	actual evapotranspiration	<input type="checkbox"/>	View result...	C:\5.Testing\NRSøby\NrSøby2003-7layer-250b\NrSøby200
5	actual transpiration	<input type="checkbox"/>	View result...	C:\5.Testing\NRSøby\NrSøby2003-7layer-250b\NrSøby200
6	actual evaporation from interception	<input type="checkbox"/>	View result...	C:\5.Testing\NRSøby\NrSøby2003-7layer-250b\NrSøby200
7	actual evaporation from ponded water	<input type="checkbox"/>	View result...	C:\5.Testing\NRSøby\NrSøby2003-7layer-250b\NrSøby200
8	canopy interception storage	<input type="checkbox"/>	View result...	C:\5.Testing\NRSøby\NrSøby2003-7layer-250b\NrSøby200
9	evapotranspiration from SZ	<input type="checkbox"/>	View result...	C:\5.Testing\NRSøby\NrSøby2003-7layer-250b\NrSøby200
10	depth of overland water	<input type="checkbox"/>	View result...	C:\5.Testing\NRSøby\NrSøby2003-7layer-250b\NrSøby200
11	overland flow in x-direction	<input type="checkbox"/>	View result...	C:\5.Testing\NRSøby\NrSøby2003-7layer-250b\NrSøby200
12	overland flow in y-direction	<input type="checkbox"/>	View result...	C:\5.Testing\NRSøby\NrSøby2003-7layer-250b\NrSøby200
13	infiltration to I17 (negative)	<input type="checkbox"/>	View result...	C:\5.Testing\NRSøby\NrSøby2003-7layer-250b\NrSøby200

This table is a list of all gridded data saved during a MIKE SHE simulation. The items in the list originate from the list of items selected in the Grid series output (V.2 p. 143) dialogue from the Setup tab.

View Result... - Clicking on the View result button will open the Results Viewer to the current item. All overlays from MIKE SHE (e.g. shape files, images, and grid files) will be transferred as overlays to the result view. However, the MIKE 11 river network is not transferred as an overlay.

Layer number for Groundwater Items - For 3D SZ data files, the layer number can be specified at the top of the table. However, the layer number can be changed from within the Results Viewer (see Changing to a different SZ layer (V.1 p. 117)) By default the top layer is displayed.

Add XY-flow Vectors - Vectors can be added to the SZ plots of results, by checking the *Add X-Y flow vectors* checkbox. These vectors are calculated based on the *Groundwater flow in X-direction* and *Groundwater flow in Y-direction* data types if they were saved during the simulation. In the current version, velocity vectors cannot be added for overland flow output.

file name - The file name column shows the name of the result file from which the gridded data will be extracted.

**Related Items:**

- Grid series output (V.2 p. 143)
- The Results viewer (V.1 p. 95)

4.2.1 “The Result Viewer setup file already exists” warning

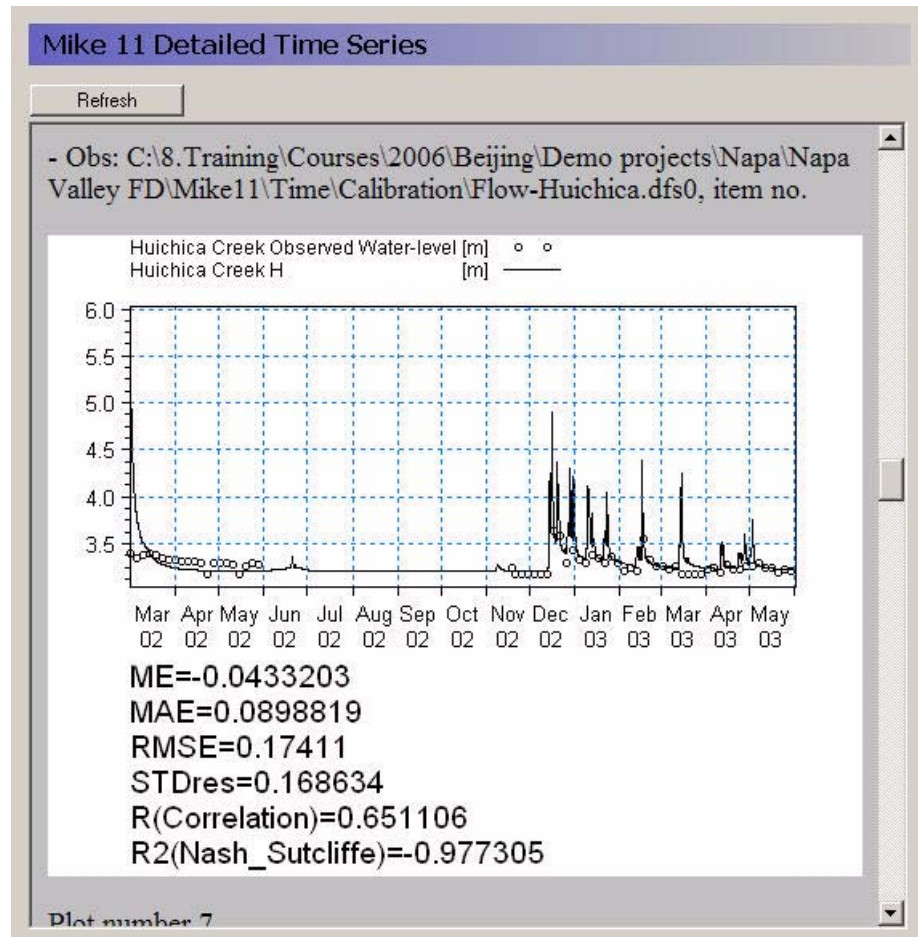
When the Result Viewer opens one of the items in the table, it creates a setup file for the particular view with the extension .rev. The name of the current setup file is displayed in the title bar of the dialogue. Initially, the .rev file includes only the default view settings and the overlay information from MIKE SHE. However, if you make changes to the view, such as changes the way contours are displayed, then when you close the view, you will be asked if you want to save your changes.

The next time you open the item in the table, you will be asked if you want to overwrite the existing .rev file. If you click on “Yes”, then a new .rev file will be created with the default values. If you click on “No”, then your previous settings will be re-loaded. This is a convenient way to set up the contouring, legend, etc., the way you want and then re-use the settings.

The .rev file can also be loaded directly in the Results Viewer by double clicking on the .rev file or loading the file into the MIKE Zero project explorer.



4.3 MIKE 11 Detailed Time Series



The MIKE 11 Detailed time series tab includes an HTML plot of each point selected in the Detailed MIKE 11 Output (V.2 p. 141) dialogue. The HTML plots are updated during the simulation whenever you enter the view. Alternatively, you can click on the Refresh button to refresh the plot.

Note that the HTML plot is regenerated every time you enter the view. So, if you have a lot of plots and a long simulation, then the regeneration can take a long time.

Related Items:

- Detailed MIKE 11 Output (V.2 p. 141)
- Statistic Calculations (V.2 p. 161)

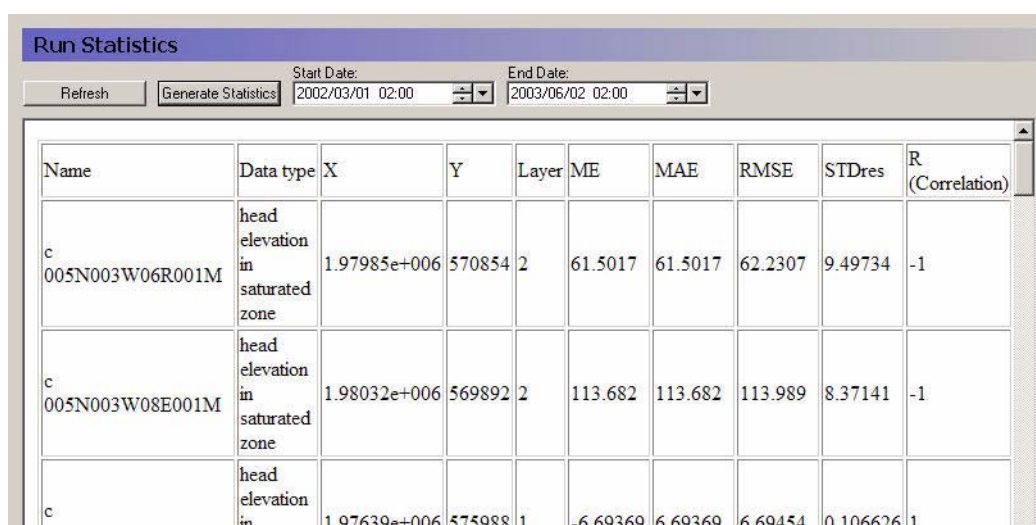


4.4 Run Statistics

Run statistics can be generated in HTML format for a MIKE SHE simulation. The run statistics table information can be copied and pasted directly into any word processing program, such as Microsoft Word, or spreadsheet, such as Microsoft Excel. The Run Statistics HTML document includes MIKE SHE and MIKE 11 results for all items included in the MIKE SHE and MIKE 11 detailed time series sections that also include observation data.

To calculate Run Statistics for a simulation, navigate to the Results Tab and the Run Statistics item on the menu tree. Press the Generate Statistics button on the Run Statistics window to perform the statistical calculations. For some simulations with long simulation periods and/or a lot of calibration data it can take several minutes to generate the run statistics, since the entire dfs0 file is loaded into memory.

After successful completion of the Generate Statistics phase, the Run Statistics HTML document will be displayed in the window on the Run Statistics page (see below).



The screenshot shows the 'Run Statistics' window with a table of results. The window has a title bar 'Run Statistics' and buttons for 'Refresh' and 'Generate Statistics'. It also displays 'Start Date: 2002/03/01 02:00' and 'End Date: 2003/06/02 02:00'. The table has columns for Name, Data type, X, Y, Layer, ME, MAE, RMSE, STDres, and R (Correlation). Three rows of data are visible, representing different simulation items.

Name	Data type	X	Y	Layer	ME	MAE	RMSE	STDres	R (Correlation)
c 005N003W06R001M	head elevation in saturated zone	1.97985e+006	570854	2	61.5017	61.5017	62.2307	9.49734	-1
c 005N003W08E001M	head elevation in saturated zone	1.98032e+006	569892	2	113.682	113.682	113.989	8.37141	-1
c -----	head elevation in	1.97639e+006	575988	1	-6.69369	6.69369	6.69454	0.106626	1

Similar to the detailed time series output, the Run Statistics can be viewed during a simulation. Press the Refresh button on the Run Statistics page to update the Run Statistics using the most recent model results during a simulation



4.4.1 Shape file output for run statistics

A shape file of statistics is also generated when the html document is generated. The shape file contains all of the information contained in the HTML document and can be used to generate maps of model errors that can be used to evaluate spatial bias. The shape file is created in the simulation directory and is named *projectname_Stat.shp* where *SimulationName* is the name of the *.she file for the simulation. Note: the Run Statistics shape file does not have a projection file associated with it and this file should be created using standard ArcGIS methods.

The statistics contained in the HTML document and the shape file are calculated using the same methods used to calculate statistics for the detailed time series output.

4.4.2 Statistic Calculations

The standard calibration statistics calculated based on the differences between the measured observations and the calculated values at the same location and time. Thus, the error, or residual, for an observation-calculation pair is

$$E_{i,t} = Obs_{i,t} - Calc_{i,t} \quad (4.1)$$

where $E_{i,t}$ is the difference between the observed and calculated values at location i and time t .

Mean (ME)

The mean error at location i where n observations exist is

$$ME_i = \bar{E}_i = \frac{\sum (Obs_{i,t} - Calc_{i,t})}{n} \quad (4.2)$$

Mean Absolute Error (MAE)

The mean absolute error at location i where n observations exist is

$$MAE_i = |\bar{E}_i| = \frac{\sum |Obs_{i,t} - Calc_{i,t}|}{n} \quad (4.3)$$

**Root Mean Square Error (RMSE)**

The root mean square error at location i where n observations exist is

$$RMSE_i = \sqrt{\frac{\sum_t (Obs_{i,t} - Calc_{i,t})^2}{n}} \quad (4.4)$$

Standard Deviation of the Residuals (STDres)

The standard deviation of the residuals at location i where n observations exist is

$$STDres_i = \sqrt{\frac{\sum_t ((Obs_{i,t} - Calc_{i,t}) - \bar{E}_i)^2}{n}} \quad (4.5)$$

Correlation Coefficient (R)

The correlation coefficient at location i is

$$r_i = \sqrt{\frac{\sum_t (Calc_{i,t} - \overline{Obs}_i)^2}{\sum_t (Obs_{i,t} - \overline{Obs}_i)^2}} \quad (4.6)$$

where \overline{Obs}_i is the mean of the observations at location i .

Nash Sutcliffe Correlation Coefficient (R2)

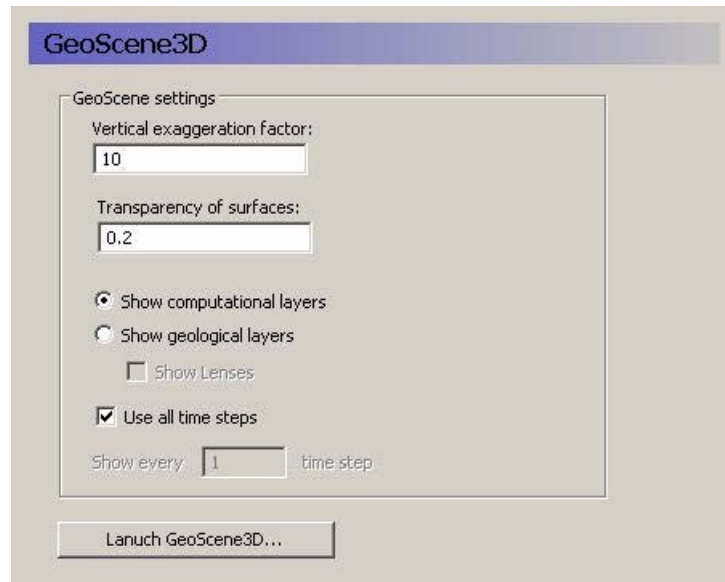
The Nash-Sutcliffe coefficient at location i where n observations exist is

$$R2 = \frac{\sum_t (Obs_{i,t} - Calc_{i,t})}{\sum_t (Obs_{i,t} - \overline{Obs}_i)} \quad (4.7)$$

where \overline{Obs}_i is the mean of the observations at location i .



4.5 GeoScene3D



The GeoScene3D dialogue provides a link to the GeoScene3D program for viewing the model results.

Vertical exaggeration factor - In most models, the lateral extent is several orders of magnitude greater than the vertical extent. Thus, without a vertical exaggeration factor, the model view will be too thin to be useful. A vertical exaggeration of 10 to 30 will typically give you a good looking model that accentuates the vertical differences.

Transparency of surfaces - The transparency factor allows you to see through the model surfaces. This gives you a better feeling of what is happening below or above the surface that you are looking at.

Show computational layers - In this case, the model's computational layers will be visible and selectable in the GeoScene3D interface.

Show geological layers - In this case, the specified conceptual geologic layers will be visible and selected in the GeoScene3D interface.

Show lenses - If you select to show lenses, then they will be displayed on top of the geologic layers.

Use all time steps - The default action is to display time varying data with all of the time steps



Show every __ time step - However, if your data files are visualized too slowly, then you can reduce the number of time steps being shown.



Phi Software

MIKE SHE EDITORS





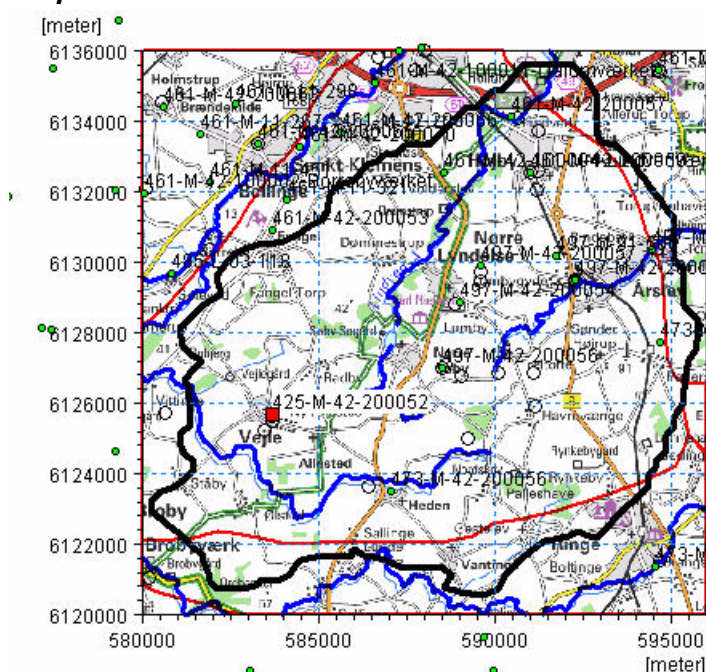
5 WELL EDITOR

The Well Editor is the MIKE SHE tool for managing pumping wells. The data file for the well editor is independent of the numerical model. That is, the well file often contains all of the wells in the entire model region, including wells outside the model area. The preprocessing takes care of including only the relevant wells in the numerical model.

The dialogue for the Well Editor is divided into

- an Interactive Map (V.2 p. 167) of well locations,
- a table of Well Locations (V.2 p. 168),
- a table of Well Filters (V.2 p. 169) for the current well, and
- a schematic Layers Display (V.2 p. 170) showing the relationship between the well screen, the current geologic model and the numerical layers.

5.0.1 Interactive Map



The interactive map displays all of the wells in the well file. Clicking on individual wells will select the corresponding item in the table of well locations. Similarly, selecting an item from the list will change the icon of the well on the map to a red square.



The overlays are automatically carried over from the model Setup Tab. You can't add or modify overlays directly in the Well Editor. This must be done from the Setup Tab

Right clicking on the map, allows you to control the zoom and a number of other functions:

Grid - turns on/off a faint coordinate grid that changes with the zoom factor

Set new area coordinates - allows you to change the displayed area of the map

Text - turns on/off the display of the Well ID labels for the wells

Export Graphic - allows you to save the view to the clipboard, or a .bmp or wmf graphic file for importing into MSWord, for example.

5.0.2 Well Locations

	Well ID	X	Y		Level	Depth	Well Field
1	425-M-42-2000	583679.00	6125678.00		0.00	0.00	modelområd
2	425-M-42-2000	579720.00	6117702.00		0.00	0.00	Undefined
3	425-M-42-2000	583040.00	6118387.00		0.00	0.00	Undefined
4	425-M-42-2000	579181.00	6124624.00		0.00	0.00	Undefined
5	427-F-07-103	598656.00	6108707.00		0.00	0.00	Undefined
6	427-M-42-1000	596254.00	6109889.00		0.00	0.00	Undefined
7	427-M-42-1000	597750.00	6108821.00		0.00	0.00	Undefined

Well_ID - This is the user specified name of the well. The Well_ID cannot contain any spaces.

X, Y - These are the X and Y map coordinates of the well.
EUM Data Units: Item geometry 2-dimensional

Level - The Level defines the maximum elevation shown on the profile view of the geologic layers, calculation layers, and screened intervals for the well. The topography is shown if the Level is less than the topography.
EUM Data Units: Elevation

Depth - The Depth is defined from the Level. It defines the maximum depth shown on the graphical view displaying the profile view of the geologic layers, calculation layers, and screened intervals for the well. The bottom of the geologic layers is shown if the Level minus the Depth is higher than the bottom of the geologic layers.
EUM Data Units: Depth below ground



Well Field - The Well Field item is used for filtering the displayed boreholes. The Mask item in the top menu bar uses the Well Field for its selection criteria.

5.0.3 Well Filters

Filter and pumping definition of selected well:

	Top	Bottom		Pumping file	Fraction
1	15.55	-11.97	<input checked="" type="checkbox"/>	C:\5.Testing\...\fyn-abs-7lag-suf_dm ... Edit New	1.00

Top - This is the elevation of the top of the screen or open hole interval for the well (in the same units (ft, m, etc.) as specified in the EUM Database for item geometry 2-dimensional).

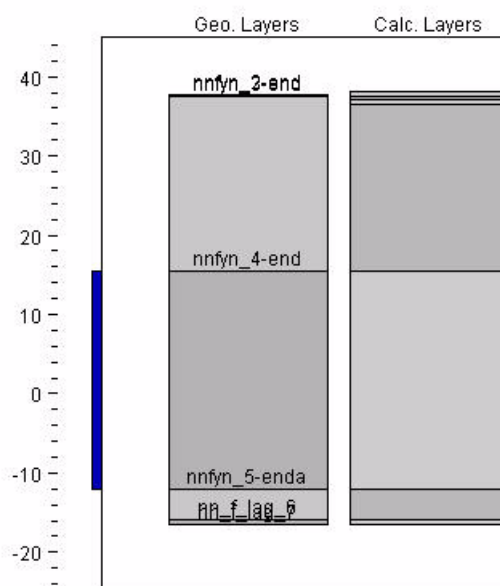
Bottom - The elevation of the bottom of the screen or open hole interval for the well (in same units (ft, m, etc.) as specified in the EUM Database for item geometry 2-dimensional).

Pumping File - Name of the .dfs0 file with groundwater pumping data for the well. When using the Browse button, [...], to select the file, you will be given the option of specifying the Item number in the .dfs0 file.

Fraction - This is a multiplier for the groundwater pumping rate specified in the DFS0 File.



5.0.4 Layers Display



The Layer display section displays the location of all well screens assigned to the well. The Geo Layers column displays the geologic layers assigned in the Setup Tab for the well, and the Calc Layers is the numerical layers for the column of model cells in which the well is located.

Both the Geo Layers and the Calc Layers items require that the model has been successfully pre-processed. If you have not pre-processed the model yet, or if during the preprocessing an error occurred, then a warning message dialogue may appear saying that the model must be pre-processed first. If this happens, the Well Editor will function normally, but the Geo and Calc layers may not be shown.

5.1 Importing Data

In the top menu bar there is an Import menu, that allows you to import the following well data.

Zeus data - The Zeus data is a specialized data format from the Geological Survey of Denmark and Greenland.

.t0 data - The .t0 format is a time series file format used by MIKE SHE prior to the MIKE Zero version. This is provided for backwards compatibility reasons.

ASCII files - This is the most common way of imported well data.



Pump definition data - This includes data related to the pump.

5.1.1 Importing data from a .t0 file

The .t0 file format was used for MIKE SHE pumping data in Release 2001 and earlier. The .t0 file format is an ASCII file format that is outlined in the pre-MIKE Zero documentation and will not be described here, but below is an example.

```

FILETYPE DATATYPE VERNO:      4      51      530
TEXTLINE          :      Groundwater abstraction
NREC DELVAL       :      10      -1E-35
START DATE        I          :      1992  1      1      0      0
END DATE          :      2000 12 31 23 59
UTM XYUNIT ZTYPE  :      32      2      1
21 555371 6138596 5 5 555371 6138596 555371 555371 'W93-3'
22 566637 6127708 7 5 566637 6127708 566637 566637 'W93-9'
23 561464 6128353 36 5 561464 6128353 561464 561464 'W94-5'
24 562873 6128089 42 5 562873 6128089 562873 562873 'W94-8'
25 563545 6140116 57 5 563545 6140116 563545 563545 'W69-2'
26 559444 6133297 27 5 559444 6133297 559444 559444 'W73-8'
27 556956 6126975 27 5 556956 6126975 556956 556956 'W74-9'
28 566406 6138802 23 3 566406 6138802 566406 566406 'W74-10'
29 567366 6137663 47 5 567366 6137663 567366 567366 'W83-6'
30 558170 6128655 16 5 558170 6128655 558170 558170 'W87-9'
1992 12 31 23 59 7 27 7 27 7 27 7 27 7 27
1993 12 31 23 59 9 25 9 25 9 25 9 25 9 25
1994 12 31 23 59 0 23 0 23 0 23 0 23 0 23
1995 12 31 23 59 6 24 6 24 6 24 6 24 6 24
1996 12 31 23 59 7 19 7 19 7 19 7 19 7 19
1997 12 31 23 59 8 19 8 19 8 19 8 19 8 19
1998 12 31 23 59 0 22 0 22 0 22 0 22 0 22
1999 12 31 23 59 0 27 0 27 0 27 0 27 0 27
2000 12 31 23 59 0 24 0 24 0 24 0 24 0 24

```

When you select this option, a file selection dialogue appears where you can select the .t0 file. If the file is properly prepared then the import should proceed automatically.

What data is imported

For each well in the .t0 file, a well is added to the list of wells including the well name and the x- and y-coordinates of the well.

The time series information for each well is read and a separate .dfs0 file is created for each of the wells in the .t0 file. These .dfs0 files are placed in a subdirectory that has the same name as the .t0 file.

Importing layer information

The layer information can only be imported if the well editor file is open at the same time as a model file (.SHE file) that has previously been pre-processed. In this case, the .t0 import utility reads the pre-processed model layer information and sets the top and bottom of the filter to the top and bottom of the specified layer in the .t0 file.



Note: This import function may not create a well file that generates identical results to pre-2002 versions of MIKE SHE if the model geometry is modified (layer elevations no longer are identical) or if wells were defined as line sinks.

Errors and warnings

Since the .t0 import utility uses FORTRAN list directed read statements, any missing information in the header will cause the import to fail. However, the import utility does not include detailed analysis of the .t0 file format, so finding the errors can be somewhat tricky.

If you encounter an error such as 'Error reading header information of T0 file' or an error related to the end of file or missing time series data, then this is often related to missing information in the t0 file. Check that all fields in the .t0 file are complete and that there are no extraneous fields. Also ensure that the well name is enclosed in quotation marks.

5.1.2 Importing *TAB delimited text file*

The most common file format to import is a TAB delimited ASCII file, typically generated from Excel or a database program.

The only restriction on the import is that only one screened interval can be specified for each well. Additional screened intervals must be specified manually after the wells have been imported.

Below is the format that each line in the ASCII file must follow:

```
Well_ID>X>Y>Level>Depth>Well_Field>Top>Bottom>Fraction>dfs0_File>dfs0_item
```

A simple example with three groundwater wells is given below.

```
CW1 7780.00 20331.00 0.00 0.00 CW 0. -60. 1. .\Time\ClassPumpage.dfs0 1
CW2 8000.00 19000.00 0.00 0.00 CW -10. -50. 0.5 .\Time\ClassPumpage.dfs0 3
CW3 7600.00 21300.00 0.00 0.00 CW 10. -60. 1. .\Time\ClassPumpage.dfs0 2
```



6 **UZ SOIL PROPERTIES EDITOR**

To solve Richards equation two important hydraulic functions are required for all soil types which characterise the individual soil profiles within the model area:

- the Soil Moisture Retention Curve and
- the Hydraulic Conductivity Function.

This information, along with the following parameters, is stored in the soil property database:

- soil moisture at saturation (θ_s) [-]
- soil moisture at effective saturation (θ_{eff}) [-]
- capillary pressure at field capacity (pF_{fc})
- capillary pressure at wilting point (pF_w)
- residual soil moisture content (θ_r) [-]
- saturated hydraulic conductivity (K_s)

pF is defined as $\log_{10}(-100\psi)$ where ψ is the matric potential. Notice that ψ is always negative under unsaturated conditions.

The soil moisture at effective saturation θ_{eff} is the maximum achievable soil moisture content.

6.1 **Soil Moisture Retention Curve**

The relationship between the water content, θ , and the matric potential, ψ , is known as the soil moisture retention curve, which is basically defined by the texture and structure of the soil. The amount and type of organic material may also have an influence on the relationship. Characteristically, the pressure head decreases rapidly as the moisture content decreases. Hysteresis is also common, that is the relationship between θ and ψ is not unique, but depends on whether the moisture content is increasing or decreasing. MIKE SHE allows for any shape of the soil moisture retention curve, but does not take hysteresis into account (i.e. a unique relation between θ and ψ is assumed).

Typically, the soil moisture curve is measured in a laboratory or assumed based on typical values for similar soils. If laboratory data is available, the measured θ - ψ values can be input directly into MIKE SHE as tabular data.



Intermediate values are then calculated by MIKE SHE, using a cubic spline method, and stored internally in the code. Alternatively, the measured values can be fitted to commonly used functional relationships. The appropriate function parameters can be input directly or more refined tabular data may be generated externally to MIKE SHE (e.g. in MS Excel) and input as tabular data.

Several parametric forms of the soil moisture retention curve have been developed over the years. The MIKE SHE interface allows the user to specify several of these parametric forms.

The pF_{fc} (field capacity) is used as the initial condition in the unsaturated flow module

The pF_w (wilting) is the lower limit at which water can be removed via evapotranspiration.

6.1.1 *Tabulated*

The data points describing the pressure conductivity curve can be given as a table of pF versus θ (moisture content) values. The table should be specified starting with the lowest value of pF (wettest condition) and given in increasing order of pF .

To get a smooth retention curve MIKE SHE adopts a cubic spline curve fitting procedure. As a minimum, you should specify the conductivity at saturation, the field capacity and the wilting point. However, this we strongly advise against this because the cubic spline function is unlikely to be able to fit an appropriate function to only 3 points.

6.1.2 *Van Genuchten and Campbell Functions*

In addition to the tabulated values, parametric functions are available using the Van Genuchten and the Campbell formulations. It is important to note that the data is tabulated internally and stored in the same form as if tabulated data were input.

6.2 *Hydraulic Conductivity Function*

The Governing Equation for the unsaturated flow requires information about two hydraulic functions: The hydraulic conductivity function, $K(\theta)$ and the soil moisture retention curve $\psi(\theta)$ are important.

The hydraulic conductivity decreases strongly as the moisture content θ decreases from saturation. This is not surprising since the total cross-sectional area for the flow decreases as the pores are getting filled with air. In



addition, when a smaller part of the pore system is available to carry the flow, the flow paths will become more tortuous. Also, there is an increase of the viscosity of the water, when the short range adsorptive forces become dominant in relation to the capillary forces.

The experimental procedure for measuring the $K(\theta)$ function is rather difficult and not very reliable. Alternatively procedures have been suggested to derive the function from more easily measurable characterizing properties of the soil or simply to rely on empirical relationships.

Reviews of various methods for predicting the conductivity function can be found in the literature.

6.2.1 *Tabulated*

The data points describing the pressure conductivity curve can be given as a table of pF versus K values. The table should be specified starting with the lowest value of pF (wettest condition) and given in increasing order of pF.

To get a smooth hydraulic conductivity curve, MIKE SHE adopts a cubic spline curve fitting procedure. As a minimum, you should specify the conductivity at saturation, the field capacity and the wilting point. However, this we strongly advise against this because the cubic spline function is unlikely to be able to fit an appropriate function to only 3 points.

6.2.2 *Averjanov*

In the Averjanov method, the hydraulic conductivity, K , is described as a function of the effective saturation, S_e :

$$K_{(E)} = K_{sat} S_e^n \quad (6.1)$$

where

$$S_e = (\theta - \theta_r) / (\theta_s - \theta_r) \quad (6.2)$$

in which θ_s , θ and θ_r are saturated, actual and residual moisture contents, respectively.

The full knowledge about the hydraulic conductivity function is seldom available, and the parameter n has to be estimated by calibration.

As a guideline the exponent n is usually small for sandy soils (2-5) and large for clayey soils (10-20). It is important to note that the value of the



exponent n will influence the percolation rate in the soil and thereby influence the actual evaporation rate.

6.2.3 Van Genuchten and Campbell/Burdine methods

In addition to the tabulated values, parametric functions are available using the Van Genuchten and the Campbell/Burdine formulations. It is important to note that the data is tabulated internally and stored in the same form as if tabulated data were input.

The van Genuchten Shape Factor, l , is soil texture dependent, with a minimum allowed value of -4.



7 *ET VEGETATION PROPERTIES EDITOR*

The vegetation editor is used to specify vegetation data for the evapotranspiration and irrigation management modules. The vegetation database contains the time varying vegetation characteristics for each type of vegetation that is specified in the model domain.

The vegetation database is optional and can be used only when the Evapotranspiration (ET) and Unsaturated Zone components are included in the model.

7.1 *Vegetation Database Items*

The vegetation database is organized around a data tree similar to the setup editor. To create a vegetation type in the database and populate with the corresponding data, simply add a vegetation item in the main dialogue and then fill out the tables in the dialogues that appear in the data tree under the new vegetation item.

7.1.1 *Specifying Vegetation Properties in a Database*

Vegetation Setup					
	Vegetation name	Vegetation Development	Include Irrigation	Evapotranspiration Parameters	Comments
				Default	Edit
1	Bare soil	User defined	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
2	HYV-Aman(160d)	User defined	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
3	HYV-Aman(120d)	User defined	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
4	Boro (145d)	User defined	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
5	Aus (120d)	User defined	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
6	Wheat	User defined	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
7	Tobacco	User defined	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
8	Grain	User defined	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
9	Potatoes	User defined	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
10	Sugarcane	User defined	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
11	Homestead	User defined	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
12	Grass	User defined	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	
13	Spring cereal (28/4)	User defined	<input type="checkbox"/>	<input checked="" type="checkbox"/>	
14	Winter Wheat	User defined	<input type="checkbox"/>	<input checked="" type="checkbox"/>	
15	Winter Barley	User defined	<input type="checkbox"/>	<input checked="" type="checkbox"/>	
16	Winter rye (25/9)	User defined	<input type="checkbox"/>	<input checked="" type="checkbox"/>	
17	Winter rane (15/9)	User defined	<input type="checkbox"/>	<input checked="" type="checkbox"/>	

The vegetation database is populated with data in a number of steps. Firstly, all the vegetation types to be included in the database are entered in a table in the Vegetation Setup Menu. The data needed are:



- Vegetation name;
- Vegetation Development;
- Include Irrigation;
- Evaporation parameters.

If the Irrigation module is used in the model you, may chose to use the irrigation demand values from the vegetation file. In this case, you should select the 'Include Irrigation' option and then specify the irrigation demand parameters by vegetation type.

You can also specify specific Evaporation Parameters for each vegetation type. If the 'Default' value is checked, then the global values defined in the main vegetation dialogue (shown below) will be used:

Vegetation

Distribution type: Global Data type: Timeseries file ET parameters ...

LAI Timeseries file: ... Edit...

RD Timeseries file: ... Edit...

If the default value is unchecked, then you can specify the ET parameters by vegetation type.

7.1.2 Vegetation stages

Stages - Wheat

	Stage name	End day	
1	1	0	
2	2	10	
3	3	40	
4	4	55	
5	5	85	
6	6	95	

The first thing to do, is to specify the standard vegetation stages for each crop in the database. These temporal variations in vegetation characteristics can normally be described by a number of characteristic stages of specific length. The changes are defined as a set of linear changes between two consecutive crop stages. Three parameters describe the stage, the leaf area index (LAI), the root depth (RD), and the crop coefficient K_c .



The day number indicates the cumulative days from crop establishment (e.g. sowing) to the end of the specific crop stage.

If in the Vegetation (V.2 p. 64) dialogue, subsequent start dates overlap with the development cycle, a warning will be issued in the log file that says the crop development was not over yet before the new crop was started. MIKE SHE will then start a new crop cycle at the new start date.

7.1.3 **Evapotranspiration Parameters**

The parameters used in the evapotranspiration calculations can be divided into three groups, which regulate interception, soil evaporation and plant transpiration, respectively.

The amount of soil water, which can be intercepted by the vegetation canopy is determined by multiplying the interception capacity, C_{int} , by the LAI. C_{int} depends on the surface characteristics of the vegetation type. The units of C_{int} are [L], but they should be interpreted as [L]/(area of leaves)/(ground area). A typical value is 0.05 mm.

The calculation of soil evaporation contains two components, the basic soil evaporation which occurs regardless of soil dryness at moisture contents in the range $\theta_W - \frac{1}{2}(\theta_W + \theta_F)$ and enhanced soil evaporation at moisture contents above $\frac{1}{2}(\theta_W + \theta_F)$. The fraction of the potential evapotranspiration, which is always allocated to the basic soil evaporation, is determined by C2. In the two-layer soil model described by Kristensen & Jensen (1975), this value was found to be 0.15. For dynamic simulation using the unsaturated zone description in MIKE SHE, a value of 0.2 was, however, found to give better results (Miljøstyrelsen, 1981; Jensen, 1983).

The transpiration from the vegetation is regulated by two parameters. C1 is the slope of the linear relation between LAI and E_a/E_p , which determines at which LAI the actual evapotranspiration equals the potential evapotranspiration at ample water supply. A typical value of C1 is 0.3. C3 regulates the influence of water stress on the transpiration process and may depend on the soil type with higher values for light soils than for heavier soils. The influence of soil dryness is reduced when C3 is increased. In Kristensen & Jensen (1975), a value of 10 mm was found for loamy soils. For simulations with the unsaturated zone description in MIKE SHE, a value of 20 mm was found more appropriate (Miljøstyrelsen, 1981; Jensen, 1983).

The root distribution in the soil is regulated by the Aroot parameter. The value of Aroot may depend on soil bulk density with higher values for soils with high bulk density where root development may be more



restricted than for soils with low bulk density. A typical value is 1 at which 60% of the root mass is located in the upper 20 cm of the soil at a root depth of 1 m. Lower Aroot values decrease this fraction and give a more even root distribution.

Note: The C1, C2, C3 ,and AROOT parameters are only used in the Richards Equation and Gravity Flow methods, and not in the 2-Layer UZ method.

7.1.4 Vegetation Development Table

User Defined Vegetation Development				
	End day	LAI	Root	Kc
1	0	2	300	1
2	10	2	300	1
3	30	4	700	1.1
4	60	5	750	1.1
5	80	5	750	1
6	90	2	750	1

For each crop stage, three vegetation parameters need to be specified:

- **LAI** - The Leaf Area Index, which is the (Area of leaves)/(Area of the ground), can vary between 0 and 7 depending of the vegetation type;
- **Root** - The Rooting Depth of the crop. It will normally vary over the season. Consideration about the soil type should be taken because some crops may develop different root distribution depending on the soil characteristics;
- **Kc** - The crop coefficient.

The leaf area index and the root depth should be specified at the end of each crop stage. The development of LAI and root depth between the specified values are then interpolated linearly by the model. In addition to these parameters, it is often necessary to supply the crop coefficient (**Kc**), which is used to adjust the reference evapotranspiration relative to the actual evapotranspiration of the specific crop.

By the FAO definition, the reference evapotranspiration represents the potential evapotranspiration for a 8-15 cm high reference grass plane with ample water supply. Most farm crops may differ from this in two ways:



- In the early crop stages, where LAI of the farm crop is lower than the LAI of the reference grass crop, the evapotranspiration of the farm crop is less than the calculated reference evapotranspiration. This is accounted for in the Kristensen & Jensen ET calculation, since a crop LAI is used as input. **Therefore, for most field crops it is therefore not necessary to specify Kc values below 1 in the early crop stages.**
- In the crop mid-season the opposite situation may occur where crop potential evapotranspiration is larger than the calculated reference evapotranspiration of the reference grass crop. This is not handled in the ET calculations, and Kc values above 1 may therefore be relevant for some crops in the mid-season during the period where crop leaf area index is at its maximum.

A Kc value of 1 means that the maximum evapotranspiration rate will equal the reference evapotranspiration rate.

If pan evaporation data are used in place of reference evapotranspiration data in the model input, it is often necessary to apply site specific pan coefficients to convert the pan evaporation to reference evapotranspiration. Pan coefficients are normally in the range 0.5 – 0.85.

7.1.5 Irrigation Parameters

In the irrigation module the amount of irrigation applied can be driven by the amount of water demanded by the crop. That is, in drier periods more irrigation water is required, so more irrigation is applied.

In the Irrigation demand dialogue, if you specify that the Vegetation Property file should be used for the demand calculations, then the demand values will be read from the vegetation file specified in the Vegetation (V.2 p. 64) dialogue.

Further, in the irrigation module, you specify the type of demand calculation:

- User specified
- Maximum allowed deficit
- Crop stress factor
- Ponding depth



In the Irrigation dialogue of the Vegetation properties file,

Irrigation							
	End day	Moisture Deficit Start	Moisture Deficit Stop	Reference	Prescribed	Stress factor	Ponding depth
1	0	0.1		Field Capacity	0	0	0
2	30	0.1	0	Field Capacity	0	0	0
3	120	0.1	0	Field Capacity	0	0	0
4	200	0.1	0	Field Capacity	0	0	0
5	255	0.1	0	Field Capacity	0	0	0
6	285	0.1	0	Field Capacity	0	0	0
7	295	0.1	0	Field Capacity	0	0	0
8	365	0.1	0	Field Capacity	0	0	0

you must specify a value for each of these demand types. Although, those that will not be used, may be left at the default values.

Maximum allowed deficit - If irrigation is handled automatically based on the actual moisture content in the soil, the soil moisture deficits are the deficits at which irrigation is going to start and stop. The soil moisture deficit is defined relative to the plant available water content in the root zone, which is the difference between a reference moisture content and the moisture content at wilting point. If, for example, the reference moisture content is the moisture content at field capacity and irrigation should start when 60 % of the available water in the root zone is used and cease when field capacity is reached, the value in the start column should be 0.6, the value in the stop column should be 0 and the reference input should be “field capacity”.

User specified - Alternatively, the irrigation amount applied in each crop stage can simply be prescribed.

Crop stress factor - The crop stress factor is the minimum allowed fraction of the reference ET that the actual ET is allowed to drop to before irrigation starts. That is, the minimum allowed $(\text{Actual ET})/(\text{Reference ET})$ relationship. This should be a value between 1 and 0.

Ponding depth - When using this option, the demand will be equal to the difference between the actual ponding depth and specified ponding depth. The option is typically used for modelling irrigation of paddy rice.



Thus:

- **Moisture Deficit Start** - the maximum allowable moisture deficit below the specified reference level. Irrigation will start at this level.
- **Moisture Deficit Stop** - This is the minimum allowable moisture deficit below the specified reference level. If irrigation takes place it will stop at this level;
- **Reference** - The reference moisture content. It can be chosen as either saturation or field capacity.
- **Prescribed** - This is the value used for the irrigation demand when 'User specified' is chosen in the Irrigation Demand (V.2 p. 76) dialogue.
- **Stress Factor** - This is the minimum allowed $(\text{Actual ET})/(\text{Reference ET})$ relationship This should be a value between 1 and 0.
- **Ponding Depth** - Irrigation will stop when the ponding depth reaches this value.

7.2 *Example database*

There is an example database (MIKE_SHE_vege.ETV) in the Examples\MIKE_SHE\Karup\ directory under the installation (MIKE SHE is by default installed in C:\Program Files\DHI\MIKEZero).

As the growing season of a given crop differs significantly depending on climatic region, the user must always adjust the vegetation development to local conditions.





8 **WATER BALANCE EDITOR**

The water balance utility is a flexible, post-processing tool for generating water balance data for MIKE SHE simulations. Output from the water balance utility can include area normalized flows (storage depths), storage changes, and model errors resulting from convergence problems. Water balance data can be generated at a variety of spatial and temporal scales and in a number of different formats.

To extract the water balance data, you must specify which simulation you are going, then specify the area of your model that you want the water balance for, and, finally, extract the MIKE SHE water balance data from the results files.

Once you have created a new water balance document, the following three tabs will be displayed

- Extraction (*V.2 p. 186*)
- Postprocessings (*V.2 p. 187*)
- Postprocessing Detail (*V.2 p. 188*)
- Results (*V.2 p. 190*)

Related items

- Using the Water Balance Tool (*V.1 p. 123*)



8.1 Extraction

Extraction

Water movement simulation

Flow result catalogue file: C:\5.Testing\MSHE projects\Qdensee\Qdense2003-hrs ...

Type of extraction

Area Type: Catchment

Resolution Type: Area

Sub-catchment grid codes

Type of input file: Dfs2 Item:

Dfs2 file: ...

Gross files

Pre-name of gross files: C:\5_1_1 ☒ Use default filename

Flow result catalogue file

A MIKE SHE simulation generates various output files depending on the options and engines selected for the MIKE SHE simulation. The .sheres file is a catalogue of all the various output files generated by the current MIKE SHE run. When you select the .sheres file, you are not specifying the particular output, but actually just a set of pointers to all the output files.

The extraction process reads all of the output files and makes itself ready to produce specific water balances. In the extraction dialogue, you specify the .sheres file for the simulation that you wish to calculate the water balance for. The .sheres file is located in the same directory as your results.

Note Although, this is an ASCII file, you should be careful not to make any changes in the file, or you may have to re-run your simulation.

Type of Extraction

You can choose to calculate the water balance on the entire model domain or in just a part of the domain. By default the calculation is for the entire domain, or catchment. If you choose the subcatchment area type, then you will be able to use a dfs2 integer grid code file to define the areas that you want individual water balances for.



If you use an area resolution, then the water balance will be a summary water balance for either the entire catchment or the sub-areas that you define.

If you use a single-cell resolution, you will be able to generate dfs2 maps of the water balance.

Sub-catchment grid codes

The subcatchment integer grid code file is only used if you have selected the sub-catchment water balance type. You can specify a delete value to exclude areas from the water balance. The grid spacing and dimensions in this dfs2 file must match exactly the model grid.

You can also specify a polygon shape file to define the sub-catchment areas. The shape file may contain multiple polygon, with the same or different codes. Further, the shape file length units do not have to be the same as the model length units (e.g. feet vs. meters).

Gross files

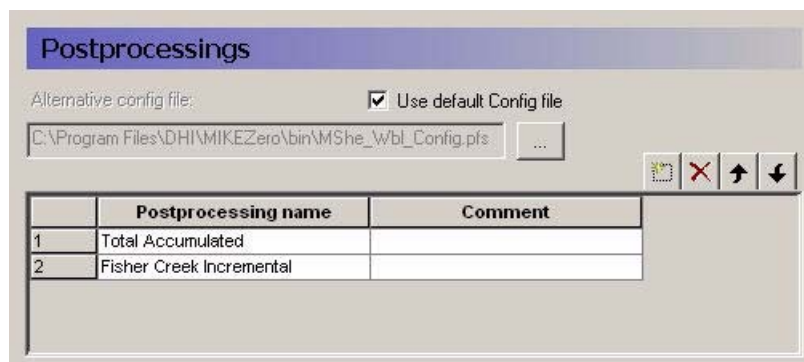
The pre-processor extracts the water balance data from the standard MIKE SHE output files and saves the data in a set of “gross” files. The file names of the gross files is built up from the project name and prefix specified here. The default value is normally fine.

Related items

- Using the Water Balance Tool (*V.1 p. 123*)

8.2 Postprocessings

After you have extracted the water balance data from the MIKE SHE results files, then you can switch to the post-processing tab. Here you can create any number of individual water balances by simply clicking on the Add item icon and specifying the water balance parameters in the parameter dialogue.



A single Postprocessing item is created by default when the water balance file is created. The default Postprocessing name can be change to a more appropriate name. Postprocessing items that are no longer needed can be deleted using the Delete button.

Use default Config file

Unchecking the Use default Config file checkbox, allows you to specify the location of a custom water balance Config file. Development of custom water balance configuration files is described in detail in Making Custom Water Balances (*V.1 p. 141*).

Related items

- Using the Water Balance Tool (*V.1 p. 123*)

8.2.1 Postprocessing Detail

For each item in the Postprocessing list above, a new item will be added to the data tree. If you expand the data tree, each will have the following dialogue.



Postprocessing

Water balance

Water balance type:

Description:

Output period

Start date: End date:

☒ Use default period

Output Timeseries Specifications

Output time step (hrs): Type:

☒ Use default output time step

Layer Output Specifications

Layer: Layer no.:

Sub-Catchment Selection

Grid code:

Single-Cell Location

X-index: Y-index:

Output File

Type: Txt file:

Water Balance

Multiple postprocessings can be run on each water balance extraction. More detail on the types of available water balances data are discussed in the Standard Water Balance Types (*V.1 p. 138*) section. In brief, the available types include

- The total water balance of the entire model catchment or sub-catchments in an ASCII table, a dfs0 file, a dfs2 map file, or a graphical chart (also by layer),
- Model errors for each hydrologic component (overland, unsaturated zone, etc.) in an ASCII table, a dfs0 file, or a dfs2 map file (also by layer),
- The snow melt and canopy/interception water balance in an ASCII table, or a dfs0 file,
- An abbreviated or detailed water balance for overland or unsaturated flow in an ASCII table, or a dfs0 file, and
- An abbreviated or detailed water balance by layer for saturated flow in an ASCII table, or a dfs0 file.



Output Period

An output period different from the total simulation period can be specified by unchecking **Use default period** and setting the **Start date** and **End date** to the period of interest

Output time series Specification

Incremental or **Accumulated** water balances can be calculated. An incremental water balance is calculated (summed) for each output time step in the Output period. An accumulated water balance each output time step is accumulated over the Output period

Layer Output Specifications

If you are using water balance types that calculate data on a layer basis, you can specify whether you want **All layers** or just the **Specified layer**, where you also must specify a layer number.

Sub-catchment Selection

If you extracted sub-catchment data from the WM results, then you must specify a subcatchment number or the name of the polygon for which you want the water balance for. The combobox contains a list of valid ID numbers or polygon names.

Single Cell Location

If you extracted the WM data by cell, then if you are not creating a map output, then you have to specify a cell location for which you want a water balance.

Output File

If you are creating a table or time series water balance, then you can write the output to either a dfs0 file or to an ASCII file for import to MSExcel, or other post-processing tool. If you are creating a map, then the output will be to a dfs2 file, with the same grid dimensions and spacing as the model grid. If you are creating a chart, then the output will be written to an ASCII file, with a special format for creating the graphic.

Related items

- Using the Water Balance Tool (*V.1 p. 123*)

8.3 Results

The data tree for the results tab lists all of the calculated water balances. The dialogue for each item, includes the file name and an Open button,



that will open an editor for the file. For ASCII output, this will be your default ASCII editor - usually Notepad. For dfs0 and dfs2 files, the DHI Time Series Editor or Grid Editor will be opened. For the chart output, the graphic will be displayed by the program WblChart.

Related items

- Using the Water Balance Tool (*V.1 p. 123*)





9 **PARTICLE TRACKING EDITOR**

MODPATH is a particle tracking program developed to work with the U. S. Geological Survey's MODFLOW code for groundwater modelling.

MODPATH has been added to MIKE SHE as an alternative to the particle tracking (PT) module in MIKE SHE. MIKE SHE's PT module is based on the random walk method, and is useful for determining dynamic well and well field capture zones, by tracking a large number of particles over time. However, the PT module can be time consuming because it is often tracking hundreds or even many thousands of particles.

MODPATH, on the other hand, is very quick as it calculates discrete path lines for individual particles. MODPATH is useful for path line and flow analysis in steady-state flow fields. In addition to computing particle paths, MODPATH keeps track of the time of travel for particles moving through the system. By carefully defining the starting locations of particles, it is possible to perform a wide range of analyses, such as delineating capture and recharge areas or drawing flow nets.

The version of MODPATH used in MIKE SHE is based on MODPATH Version 3, downloaded directly from the USGS web site.

At the moment, only steady-state, MIKE SHE SZ flow fields are supported. In other words, the particle tracking is calculated based on the flow field from one MIKE SHE time step.

MODPATH requires that the MIKE SHE result files include

- head elevation in the saturated zone,
- groundwater flow in the x-direction,
- groundwater flow in the y-direction, and
- groundwater flow in the z-direction.

MODPATH Output

MODPATH outputs the standard output files, as well as two shape files. The point theme shape file, *projectname_point.shp*, contains the particle locations at each storing time step plus the end point. The line theme shape file, *projectname_line.shp*, contains the path line for each particle calculated at each storing time step plus the end point.



9.1 Running MODPATH outside of MIKE SHE

MODPATH can be run from the user interface, or from a DOS command line or batch file.

If you are running the program from the command line, you need to know about the following executables:

- ModPathDiagPFS.exe - The main executable.
- ModPathConv.exe - Handles the conversion between MIKE SHE and MODPATH.
- ModPath3.exe - Standard USGS version of MODPATH

The ModPathDiagPFS.exe file is called using an ASCII input file using the .pfs format. The input file can be used as an argument when calling the executable. If you do not include the file name as an argument, then a browse dialogue will appear when you launch the executable.

Note - the current set of executables does not include the APV functions.

The format of the input file is shown below.

Note - This file is automatically generated from the GUI.

Line item	Comment
// Created : 2006-12-6 13:13:55 // DLL id : C:\PROGRA~1\COMMON~1\DHI\MikeZero\pfs2004.dll // PFS version : Dec 4 2006 20:53:34 [MODPATH_Parameters]	The first 5 lines are header lines and should not be modified
PointLocation = .\test_PTPos1.txt	Path and file name for the initial particle locations
SheresFile = .\KarupForAD.sheres	Path to the catalogue file for the MIKE SHE results
ResultName = 'test_PTRes1'	File name suffix for the output files
Start = '10-02-1981'	The start date must be within the WM flow simulation period. If the start date does not match a saved storing time step, then the previous stored time step will be used. The flow field from this time step will be used as the steady-state flow field by MODPATH.



Line item	Comment
End = '01-02-1983 10:57:16'	MODPATH will calculate particle locations until the end date, or until all particles are removed by sinks.
StoringFrequency = 0.5	The frequency for storing particle locations [Years]
SinkStrength = 0.1	Strength of sinks in MODPATH. Must be between 0 and 1. 1: all inflow to the cell are captured by sinks in the cell (e.g. wells or other boundary conditions) 0: particles are not captured
PathDir = 0 EndSect // MODPATH_Parameters	Particle tracking direction 0: forward tracking 1: backward tracking

Initial particle locations

The initial particle locations can be defined by either a space-delimited ASCII text file or a GIS shape file.

An ASCII text file must have the following format

```
X-coordinate Y-coordinate Layer Pos
```

where the X and Y coordinates are in the same model coordinates as the flow model, the Layer is the numerical layer number of the model starting with 1 at the top, and the Pos is the relative vertical position in the layer with 1 meaning that the particle is at the top of the layer and 0 meaning that the particle is at the bottom of the layer.

A GIS shape file must contain a Layer field and a Pos field, where these have the same meaning as in the ASCII file.

9.2 Using MODPATH in the MIKE SHE GUI

The MODPATH user interface is opened by selecting New\File... in the top menu, or by clicking on the New icon in the icon bar, and then selecting **MShe Particle Tracking (.trpt)** from the MIKE SHE document list.

The MIKE SHE Particle Tracking user interface follows the same format as the other MIKE Zero user interfaces with three tabs - a Setup Tab, a Simulation Tab and a Results Tab.



9.3 Setup Tab

The Setup tab is used to define the basic file information for MODPATH, including the overlays and the MIKE SHE result files that should be used.

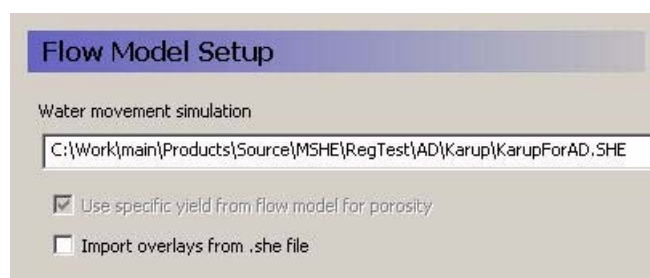
9.3.1 Display

The display items are automatically carried over from the MIKE SHE Setup. The Display dialogues are exactly the same controls as those used in the MIKE SHE Setup Tab.

Related items

- **Display (V.2 p. 20)** in MIKE SHE Setup

9.3.2 Flow model setup



You must specify a MIKE SHE water movement (.SHE) simulation that has been run. The MIKE SHE output must include the

- head elevation in the saturated zone,
- groundwater flow in the x-direction,
- groundwater flow in the y-direction, and
- groundwater flow in the z-direction.

Use specific yield from flow model for porosity - The flow velocity is controlled by the effective porosity. However, a standard MIKE SHE flow does not include the effective porosity as a parameter. The Specific yield is a reasonable approximation for the effective porosity in most cases. In the current version, MODPATH will always use the specific yield for effective porosity.

Import overlays from .she file - By default, the MIKE SHE overlays will be displayed in the Particle Tracking Editor, but this can be disabled by unchecking this checkbox.



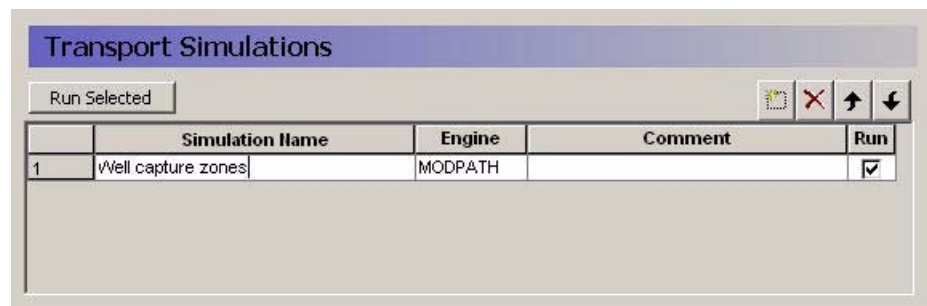
Specify the MIKE SHE setup file to use for the simulations. The GUI automatically located the result file based on the input files.

The check box “Import overlays from she file” should be checked on if all the overlays from the she files is to be imported into the MODPATH for MIKE SHE setup.

9.4 Simulation Tab

The Simulation tab is used to define the various particle tracking simulations that you want to run on the specified WM flow results.

9.4.1 Transport Simulations



You can run any number of particle tracking simulations on the flow model results. This allows you to set up standard particle tracking simulations and then run several water movement scenarios.

Simulation Name - An identifying name for the particle tracking simulation.

Engine - In the current version, MODPATH is the only available engine. MIKE SHE’s particle tracking (PT) module will be added here in a later version.

Comment - Supplementary information on the particle tracking simulation.

Run and Run Selected - the Run Selected button will run all of the particle tracking simulations that have a check in the Run checkbox.



9.4.2 MODPATH simulation specification

Well capture zones

Start: 10-02-1981

End: 01-02-1983 10:57:16

Storing frequency: 0.5

Sink strength: 0.1

Pathline direction:

☒ Forward

☐ Backward

For each MODPATH simulation defined in the Transport Simulations dialogue, the control parameters for the MODPATH simulation must be specified.

Start date - This is the both the start date for the MODPATH simulation and the saved time step date from the WM simulation to be used as the steady-state flow field. Thus, the Start date must be within the WM simulation period. The start date format follows the format defined in your Regional Settings for Windows.

End date - MODPATH will terminate when the End date is reached or when all of the particles have been removed by sinks. The end date format follows the format defined in your Regional Settings for Windows.

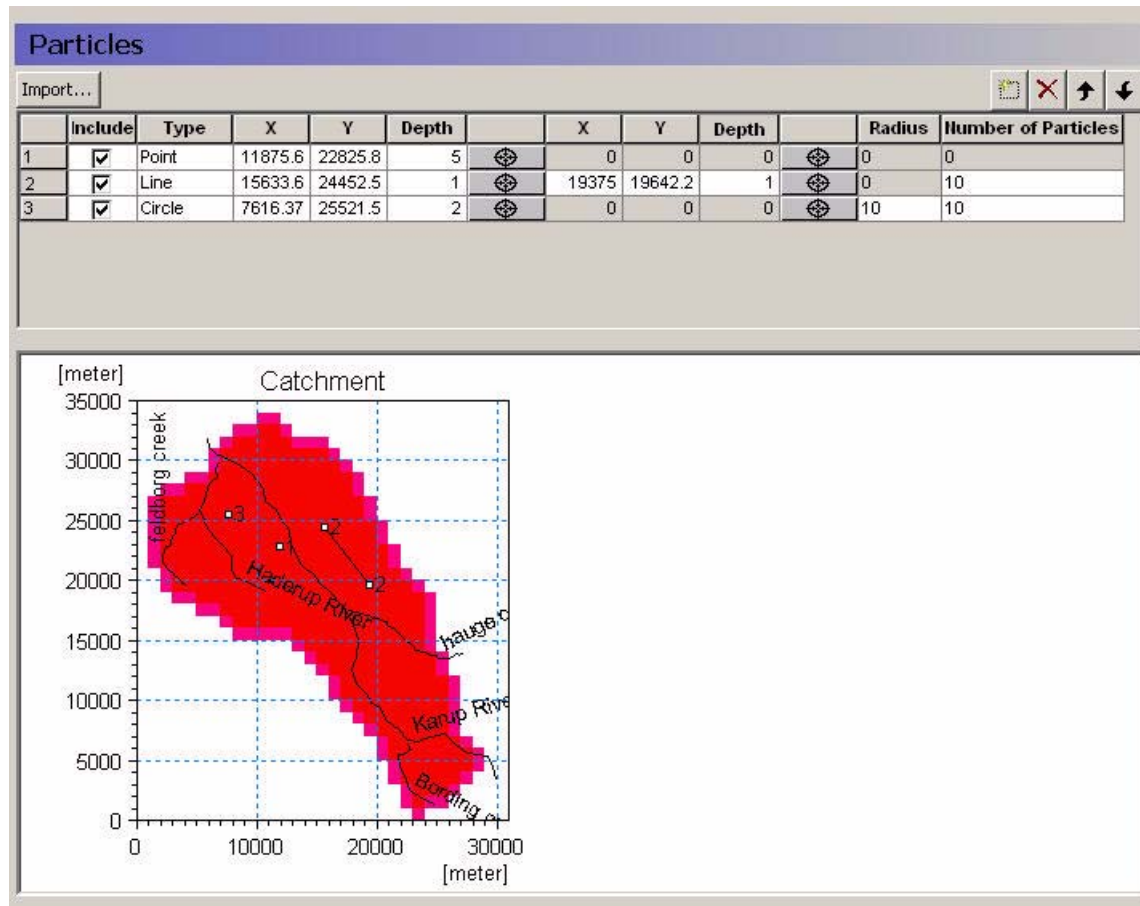
Storing frequency - MODPATH will store the particle locations at this frequency. The storing frequency must be input in [Years]. This value has not yet been connected to the EUM system.

Sink strength - The sink strength controls how strongly boundary conditions that remove water will also remove particles. A sink strength of 1 means that all particles will be removed when they enter the cell. A sink strength of 0 means than none will be removed. For more information on the Sink Strength parameter, please refer to the MODPATH documentation.

Path Line direction - MODPATH can track the particle path lines forward in time or backward in time. In other words, the initial particle location can be either a starting point or an ending point for the path line.



9.4.3 Particles



The initial particle locations can be defined by points, lines, and circles. You can specify any number of particles in this dialogue and all of them will be tracked in the MODPATH simulation.

Include - This checkbox allows you to exclude some particles from the simulation without actually removing them from the list.

Type - You can define single particles, a line of particles, or a circle of particles. Single particles are useful for evaluating the flow direction in areas of interest. Lines are useful for determining the influence of boundary conditions. Circles are useful for calculating well capture zones.

X and Y - The X and Y coordinates can be typed in by hand, or you can use the “target” button to interactively locate the particles on the map. The second set of X and Y coordinates specify the end point location for particle lines.



Depth - The depth is depth below ground for the particle location. This is translated into a model layer before MODPATH is run. In the case of particle lines, the start point and the end point can be located at different depths.

Layer - The layer option allows you to specify the model layer for the particle.

Radius - In the case of particle circles, the radius specifies the radius of the circle. The radius should normally be greater than the cell size. Otherwise, all of the particles could be located in the same cell which will not give you a realistic well capture zone.

Number of particles - This is the number of particles along the line or around the circle. The particles are evenly distributed along the line or circumference.

Import

An set of single particles can be imported to the table using either an ASCII text file or a point-theme shape file. The ASCII text file must be space delimited with the following fields, without a header

```
X_coord Y_coord Depth Layer
```

The shape file must be defined as a point-theme with either a Depth or Layer attribute. If both are defined, then both will be imported but the Layer option will control.

9.5 Results Tab

In the Results tab a data item is created for each of the Particle Tracking Simulations specified in the Simulation Tab (*V.2 p. 197*).

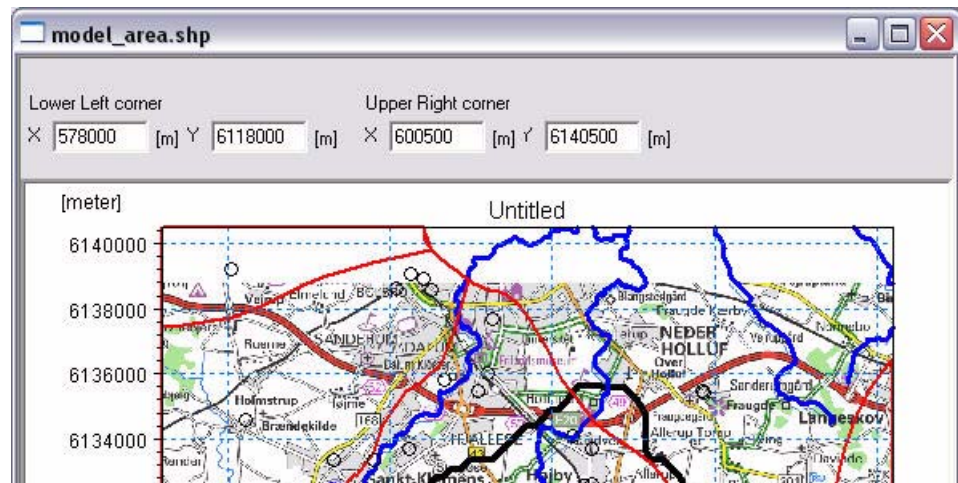
MODPATH Results

MODPATH generates a number of files, which are described in the MODPATH documentation.

In addition to the standard files, we have modified the MODPATH code to also output two shape files - a point-theme file with all of the particle locations at every save time, and a line-theme file with all of the path lines. The map view adds these two shape files to the other overlays defined in the Display (*V.2 p. 196*) dialogue.



10 SIMPLE SHAPE EDITOR



The simple shape editor is a tool for creating and editing .shp file polygons. Polygon .shp files can be used in many places in MIKE SHE, but the Simple Shape Editor is primarily used for creating and editing single discrete polygons, such as for the model domain. However, multiple polygons can be created and edited in the tool.

At the moment, the Simple Shape Editor can be launched directly from only two dialogues - the Model Domain and Grid, and the Saturated Zone Internal Boundary Conditions. If you launch it from one of these two dialogues, all of the active background maps will be transferred to and displayed the editor. However, if you want to change any of the display settings or add or subtract overlays, you must exit the editor and make the changes in the Display data item in the Setup Tab.

It is also possible to launch the editor from the New and Open menu functions, but in this case none of the background maps will be added and you cannot add maps within the tool itself. However, if you want to use the tool for creating .shp file maps for functions outside of the Model Domain and Internal Boundary Conditions, then you can still do this by using the New function on one of these dialogues, and then creating and editing the polygon normally. Then simply loading the .shp file in a different dialogue.

The Editor consists of three parts - the map coordinate text boxes, the interactive map view and the toolbar in the top menu.

Map Coordinates


The map coordinates are used to change the displayed extents of the map view. Thus, if you want to create a polygon larger or much smaller than





the current map view in the Setup Tab, you can change the coordinates of the lower left hand corner or the upper right hand corner of the map view.


Interactive Map


The interactive map is used to display the polygon or polygons that your are editing. You can use the icons in the toolbar to add, move, delete, rename etc. the polygons in the map view. The following is a description of the icons.


 - **Create Polygon** - To create a polygon, click on this icon, then left click once on the map and trace the polygon on the map that you want to create while holding down the left mouse button. This will create a series of points along the line that you trace. If you want to stop, simply lift the left mouse button and reposition the mouse when you want to start again. To finish and close the polygon, double click the left mouse button.

 - **Select/Move Polygon** - To select a polygon, click on this icon and then click on the polygon that you want to select. This will make all of the nodes on the polygon visible by adding node markers around the polygon. To move the entire polygon, with the mouse inside the polygon, left click on the polygon and while holding down the mouse drag the polygon around on the screen.

 - **Add Point** - To add a point to a selected polygon, click on this icon and then left click on the polygon where you want to add a point. A new point will be added on the line.

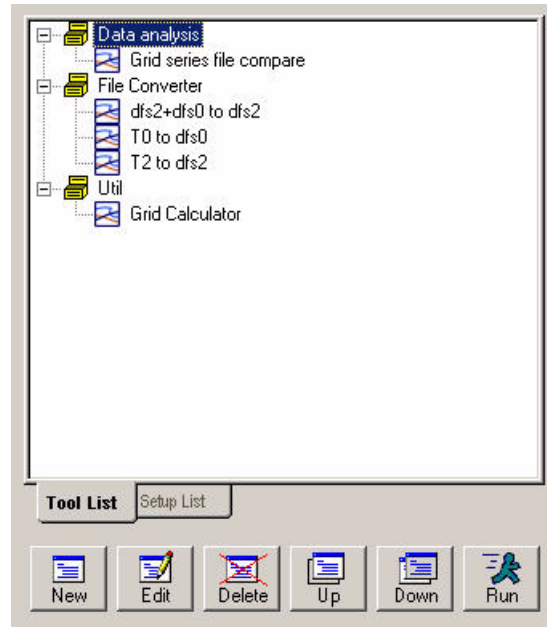
 - **Move Point** - To move a point in a selected polygon, click on this icon, then left on the point you want to move and drag the point to its new location, while holding down the mouse.

 - **Erase Point/Polygon** - To erase a point in a selected polygon, click on this icon and then click on the point that you want to delete. To delete an entire polygon, double click on the selected polygon.

 - **Add name** - to add or modify the name of a polygon, click on this icon and then left click on the polygon. A small dialogue will appear where you can change the name.



11 MIKE SHE TOOLBOX



The MIKE SHE tool box contains several wizard-like tools with Next and Back buttons to move between dialogues. In the first dialogue you specify a name for the setup. If you save the setup, then this name will be available in the Toolbox menu, allowing you to create custom set ups for your project.

The last Next button will take you to a summary dialogue, where you can execute the function, by clicking on the Execute button.

To exit the wizard, click on the Finish button, which will temporarily save your setup. If you click Cancel your setup will not be saved. After clicking Finish, you must click the Save icon in the top menu bar to permanently save your setup.

11.1 Data Analysis

11.1.1 Grid series file compare

The Grid series file compare tool is very useful during calibration and for scenario analysis. With this tool, you can do create a composite grid series file between two dfs2 or dfs3 output files.



After the initial Tool Name dialogue, is the calculation setup dialogue.

11.2 File Converter

11.2.1 dfs2+dfs0 to dfs2

This tool is used to build a time varying dfs2 file from a dfs2 grid code file and one or more dfs0 time series files. The time varying dfs2 file can be used for time varying gridded data items, including Precipitation Rate and Evapotranspiration.

As with all the Toolbox tools, the tool runs silently. If there are any errors, then the tool will simply not run.

The dfs2 file must have an EUM type of Grid Codes with a unit of Integers. See EUM Data Units (*V.1 p. 271*).

For Precipitation Rate (*V.2 p. 58*) data, the dfs0 files must be Mean Step Accumulated (*V.1 p. 247*) with an EUM Type of Precipitation Rate, and any valid unit, such as [inches/hour] or [mm/day].

item



11.2.2 *t0 to dfs0 and t2 to dfs2*

These two tools are used for converting the t0 and t2 data file formats that were used in MIKE SHE up to and including the 2001 Release.

11.3 *Util*

11.3.1 *Grid calculator*

The Grid Calculator tool allows you to perform complex operations on .dfs2 grid files. However, the grid files must have the same grid dimensions and they may not include multiple time steps or multiple items. Thus, this tool is much more restrictive than the grid operations available in the Grid Editor. However, you can make complex chains of operations and save the setup, which can save you a lot of time if you are doing the same operation many times or after each simulation.

After the initial Tool Name dialogue, is the calculation setup dialogue.

	Variable Name	Data File
1	A	C:\Maps\Topography.dfs2
2	B	C:\Maps\Lower Level.dfs2
3		
4		
5		
6		
7		
8		

Output specification:

Description: Thickness of Layer 1

File name: C:\Maps\Thickness.dfs2

Item type: Layer Thickness Item unit: meter

Expression: A-B

Example: (Level1 - Level2) * Conduc

< Back Next > Cancel Help

In the first part of this dialogue, you need to specify the files and variable names used in the mathematical operation.

In the second part, you must specify the output file name (including the .dfs2 file extension), as well as the EUM unit type and units for the resulting file (see EUM Data Units). Although the EUM list is alphabetical, the list of EUM data types is very long, as it includes all of the available EUM types for all of the MIKE Zero products. If you are trying to create a particular input type for MIKE SHE, then you should look in the MIKE SHE Setup dialogue, to find out first what EUM data type is required and then find this type in the list.



Note all fields must be filled in.

The last part of the dialogue is the mathematical expression you wish to evaluate. Many common mathematical operations are available, including LOG and LOG10. If the operation is not recognized, then an error will be generated when you try to run the expression. The standard order of operations is followed, including the use of nested brackets.

Note all variable names are case sensitive

When you are finished, the Next button will take you to a summary dialogue, where you can execute the function, by clicking on the Execute button. To exit the wizard, click on the Finish button, which will temporarily save your setup. If you click Cancel your setup will not be saved.

After clicking Finish, you must click the Save icon in the top menu bar to permanently save your setup.



Phi Software

**TECHNICAL REFERENCE FOR
WATER MOVEMENT**





12 WATER MOVEMENT OVERVIEW

This section includes detailed descriptions of the numeric engines used for moving water in MIKE SHE, including

- Overland Flow - Reference
 - Finite Difference Method
 - Simplified Overland Flow Routing
- Channel Flow - Reference
- Evapotranspiration - Reference
 - Kristensen and Jensen method
 - Simplified ET for the Two-Layer Water Balance Method
- Unsaturated Flow - Reference
 - Richards Equation
 - Gravity Flow
 - Two-Layer Water Balance
- Saturated Flow - Reference
 - 3D Finite Difference Method
 - Linear Reservoir Method





13 OVERLAND FLOW - REFERENCE

When the net rainfall rate exceeds the infiltration capacity of the soil, water is ponded on the ground surface. This water is available as surface runoff, to be routed downhill towards the river system. The exact route and quantity is determined by the topography and flow resistance, as well as the losses due to evaporation and infiltration along the flow path.

The water flow on the ground surface is calculated by MIKE SHE's Overland Flow Module, using the diffusive wave approximation of the Saint Venant equations, or using a semi-distributed approach based on the Mannings equation.

This chapter is the technical reference for the Overland Flow Module in MIKE SHE.

13.1 Finite Difference Method

13.1.1 Diffusive Wave Approximation

Using rectangular Cartesian (x, y) coordinates in the horizontal plane, let the ground surface level be $z_g(x, y)$, the flow depth (above the ground surface) be $h(x, y)$, and the flow velocities in the x - and y -directions be $u(x, y)$ and $v(x, y)$ respectively. Let $i(x, y)$ be the net input into overland flow (net rainfall less infiltration). Then the conservation of mass gives

$$\frac{\partial h}{\partial t} + \frac{\partial}{\partial x}(uh) + \frac{\partial}{\partial y}(vh) = i \quad (13.1)$$

and the momentum equation gives

$$S_{fx} = S_{Ox} - \frac{\partial h}{\partial x} - \frac{u}{g} \frac{\partial u}{\partial x} - \frac{1}{g} \frac{\partial u}{\partial t} - \frac{qu}{gh} \quad (13.2a)$$

$$S_{fy} = S_{Oy} - \frac{\partial h}{\partial y} - \frac{v}{g} \frac{\partial v}{\partial y} - \frac{1}{g} \frac{\partial v}{\partial t} - \frac{qv}{gh} \quad (13.2b)$$

where S_f is the friction slopes in the x - and y -directions and S_O is the slope of the ground surface. Equations (13.1), (13.2a) and (13.2b) are known as the St. Venant equations and when solved yield a fully dynamic description of shallow, (two-dimensional) free surface flow.



The dynamic solution of the two-dimensional St. Venant equations is numerically challenging. Therefore, it is common to reduce the complexity of the problem by dropping the last three terms of the momentum equation. Thereby, we are ignoring momentum losses due to local and convective acceleration and lateral inflows perpendicular to the flow direction. This is known as the diffusive wave approximation, which is implemented in MIKE SHE.

Considering only flow in the x-direction the diffusive wave approximation is

$$S_{fx} = S_{Ox} - \frac{\partial h}{\partial x} = -\frac{\partial z_g}{\partial x} - \frac{\partial h}{\partial x} \quad (13.3)$$

If we further simplify Equation (13.3) using the relationship $z = z_g + h$ it reduces to

$$S_{fx} = -\frac{\partial}{\partial x}(z_g + h) = -\frac{\partial z}{\partial x} \quad (13.4)$$

in the x-direction. In the y-direction Equation (13.4) becomes

$$S_{fy} = -\frac{\partial}{\partial y}(z_g + h) = -\frac{\partial z}{\partial y} \quad (13.5)$$

Use of the diffusive wave approximation allows the depth of flow to vary significantly between neighbouring cells and backwater conditions to be simulated. However, as with any numerical solution of non-linear differential equations numerical problems can occur when the slope of the water surface profile is very shallow and the velocities are very low.

Now, if a Strickler/Manning-type law for each friction slope is used; with Strickler coefficients K_x and K_y in the two directions, then

$$S_{fx} = \frac{u^2}{K_x^2 h^{4/3}} \quad (13.6a)$$

$$S_{fy} = \frac{v^2}{K_y^2 h^{4/3}} \quad (13.6b)$$



Substituting Equations (13.4) and (13.5) into Equations (13.6a) and (13.6b) results in

$$\frac{u^2}{K_x^2 h^{4/3}} = -\frac{\partial z}{\partial x} \quad (13.7a)$$

$$\frac{v^2}{K_y^2 h^{4/3}} = -\frac{\partial z}{\partial y} \quad (13.7b)$$

After simplifying Equations (13.7a) and (13.7b) and multiply both sides of the equations by h , the relationship between the velocities and the depths may be written as

$$uh = K_x \left(-\frac{\partial z}{\partial x} \right)^{1/2} h^{5/3} \quad (13.8a)$$

$$vh = K_y \left(-\frac{\partial z}{\partial y} \right)^{1/2} h^{5/3} \quad (13.8b)$$

Note that the quantities uh and vh represent discharge per unit length along the cell boundary, in the x- and y-directions respectively.

Also note that the **Stickler roughness coefficient** is equivalent to the **Manning M**. The Manning M is the inverse of the commonly used **Mannings n**. The value of **n** is typically in the range of 0.01 (smooth channels) to 0.10 (thickly vegetated channels), which correspond to values of **M** between 100 and 10, respectively.



13.1.2 Finite Difference Formulation

Consider the overland flow in a small region (see Figure 13.1) of a MIKE SHE model, having sides of length Δx and Δy and a water depth of $h(t)$ at time t .

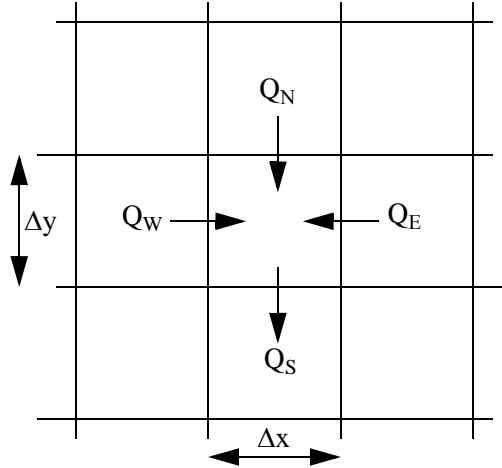


Figure 13.1 Square Grid System in a small Region of a MIKE SHE model

A finite-difference form of the velocity terms in Eq. (13.1) may be derived from the approximations

$$\frac{\partial}{\partial x}(uh) \cong \frac{1}{\Delta x} \{ (uh)_{east} - (uh)_{west} \} \quad (13.9)$$

and

$$\frac{\partial}{\partial y}(vh) \cong \frac{1}{\Delta y} \{ (vh)_{north} - (vh)_{south} \} \quad (13.10)$$

where the subscripts denote the evaluation of the quantity on the appropriate side of the square, and noting that, for example, $\Delta x (uh)_{west}$ is the volume flow across the western boundary

$$\Delta h = h(t + \Delta t) - h(t) = I + \frac{\Sigma Q \Delta t}{\Delta x^2} \quad (13.11)$$



where

$$I = i\Delta x^2, \quad (13.12)$$

$$\Sigma Q = Q_N + Q_S + Q_E + Q_W$$

and where, i is the net input to overland flow in Eq. (13.1) and the Q 's are the flows into the square across its north, south, east and west boundaries evaluated at time t .

Now consider the flow across any boundary between squares (see Figure 13.2), where Z_U and Z_D are the higher and lower of the two water levels referred to datum. Let the depth of water in the square corresponding to Z_U be h_U and that in the square corresponding to Z_D to h_D .

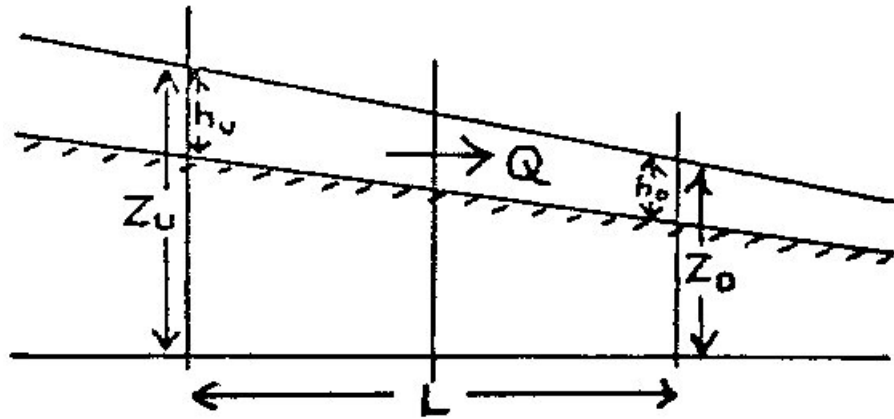


Figure 13.2 Overland flow across grid square boundary.

Equations (13.8a) and (13.8b) may be used to estimate the flow, Q , between grid squares by

$$Q = \frac{K\Delta x}{\Delta x^{1/2}}(Z_U - Z_D)^{1/2} h_u^{5/3} \quad (13.13)$$

where K is the appropriate Strickler coefficient and the water depth, h_u , is the depth of water that can freely flow into the next cell. This depth is equal to the actual water depth minus detention storage, since detention storage is ponded water that is trapped in shallow surface depressions. Equation (13.13) also implies that the overland flow into the cell will be zero if the upstream depth is zero. The flow across open boundaries at the edge of the model is also calculated with Eq. (13.13), using the specified boundary water levels.



13.1.3 Successive Over-Relaxation (SOR) Numerical Solution

The method for solving the overland flow equations is similar to the method applied to the saturated zone flow. A linear matrix system of N equations with N unknown water levels is derived. The matrix is then solved iteratively, using the modified Gauss Seidel method. Because of the non-linear relationship between water levels and flows, the 2nd order term is included in the Taylor series expressing the correction of water levels as a function of the residuals.

The flow is calculated for the remainder of any iteration using Eq. (13.13) whenever there is sufficient water in a cell, that is, whenever h_u exceeds the minimum threshold that is specified by the user.

The exchange between ponded water and the other hydrologic components (e.g. direct exchange with the saturated zone, unsaturated infiltration, and evaporation) is added or subtracted from the amount of ponded water in the cell at the beginning of every overland flow time step.

Water balance correction

As the flow equations, so to speak, are explicit during one iteration, it is necessary to reduce the calculated flows in some situations to avoid internal water balance errors and divergence of the solution scheme.

Thus, requiring that the water depth cannot be negative, which implies that $\Delta h \geq -h(t)$, rearranging Eq. (13.11) gives:

$$\Sigma Q \geq \frac{-\Delta x^2 h(t)}{\Delta t} - I \quad (13.14)$$

where ΣQ is the sum of outflows and inflows.

Splitting ΣQ into inflows and outflows and remembering that outflow is negative, gives:

$$\Sigma |Q_{out}| \leq \Sigma Q_{in} + I + \frac{\Delta x^2 h(t)}{\Delta t} \quad (13.15)$$

remembering that $I = i\Delta x^2$ and i is the net input into overland flow (net rainfall less infiltration).

If necessary during an iteration, these calculated outflows are reduced to satisfy the equal sign of (13.15).



To ensure that the inflows, ΣQ_{in} , have been summed before calculating ΣQ_{out} , the grid squares are treated in order of descending ground levels during each iteration.

13.1.4 Explicit Numerical Solution

The explicit solution is different from the SOR solution in the sense that there is no iterative matrix solution. In other words, the exchange flows between every cell and to the river are simply calculated based on the individual cell heads. However, the explicit solution is much more restrictive in terms of time step. For the explicit solution to be stable, the flow must be slow relative to the time step. For example, a flood wave cannot cross a cell in one time step. So, this leads to the following 3-step calculation process for the explicit solution of the overland flow:

- 1 Calculate all flow rates and discharges between cells and between the overland cells and river links based on the current water levels
- 2 Loop over all the cells and calculate the maximum allowed time step length for the current time step, based on the following criteria
 - Courant criteria (see next section)
 - Cell volume criteria - the volume in the cell divided by the flow rate
 - River link volume criteria - the volume in the river link divided by the flow rate
 - River bank criteria - the exchangeable volume in the river link based on the river bank elevation divided by the flow rate

In most cases, the Courant criteria is the critical criteria for the maximum time step, with the Cell volume criteria sometimes being critical. The River link and River bank criteria are less commonly critical, but may become critical when the river is very shallow.

- 3 Calculate the actual flows between the all the cells and to/from the river links using the maximum allowed time step and update all the cell water depths.

Courant criteria

Courant number, C , represents the ratio of physical wave speed to the 'grid speed' and is calculated as

$$C = \frac{\frac{dQ}{dA}}{\frac{dx}{dt}} = \frac{1}{dx} \times \frac{dQ}{dh} \times \frac{dt}{dx} \quad (13.16)$$



where do is the change in flow rate, dA is the change in cross-sectional area and oh is the change in water level.

For a stable explicit solution the courant number must be less than one.

Solving for the time step, yields,

$$\Delta t = \frac{dh}{dQ} \cdot \Delta x^2 \cdot C \quad (13.17)$$

where Δt is the time step and Δx is the cell width.

The differential term in (13.17) is the inverse of the derivative of the Mannings equation, (13.13), with respect to h , which goes to zero as the gradient approaches zero. Thus, very low gradients, for example in very flat areas, will require very small time steps. Likewise, smaller grid spacing will also lead to smaller time steps.

13.1.5 Boundary conditions

The outer boundary condition for the overland flow solver is a specified head, based on the initial water depth in the outer nodes of the model domain. Thus, if the water depth inside the model domain is greater than the initial depth on the boundary, water will flow out of the model. If the water depth is less than the initial depth on the boundary, the boundary will act as a source of water.

13.1.6 Low gradient damping function

In flat areas with ponded water, the head gradient between grid cells will be zero or nearly zero and numerical instabilities will be likely. To dampen these numerical instabilities in areas with low lateral gradients the calculated intercell flows are multiplied by a damping function. Essentially, the damping function slows down the flow between cells. You can think of the damping function as a function that increases the resistance to flow as the gradient goes to zero. This makes the solution more stable and allows for larger time steps. However, the resulting gradients will be artificially high in the affected cells and the solution will begin to diverge from the Man-



nings solution. At very low gradients this is normally insignificant, but as the gradient increases the differences can become noticeable.

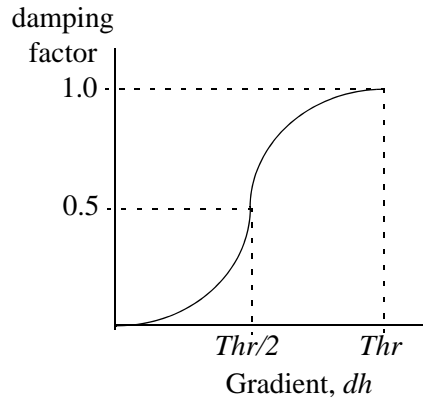


Figure 13.3 Damping function for numerical stability at low gradients

The damping function is controlled by a threshold gradient below which the damping becomes active. The actual damping function is a pair of parabolic equations. When the gradient reaches the threshold the following damping function is applied

$$F_D = 1 - 2\left(\frac{Thr - dh}{Thr}\right)^2 \quad (13.18)$$

where Thr is the threshold value and dh is the gradient. When the gradient reaches $Thr/2$, the damping function changes to

$$F_D = 2\left(\frac{dh}{Thr}\right)^2 \quad (13.19)$$

which goes to zero as the gradient goes to zero.

You can get reasonable results with a threshold gradient between 0.0005 and 0.001. Higher values may lead to a divergence from the Mannings solution. Lower values may lead to more accurate solutions, but at the expense of numerical instabilities, smaller time steps and longer simulation times.



13.2 Simplified Overland Flow Routing

The conceptual reservoir representation of overland flow in MIKE SHE is based on an empirical relation between flow depth and surface detention, together with the Manning equation describing the discharge under turbulent flow conditions (Crawford and Linsley, 1966). This was implemented in the Stanford watershed model and in its descendants, such as HSPF (Donigian et al., 1995), and has been applied in other codes such as the WATBAL model (Knudsen, J. 1985a,b; Refsgaard and Knudsen, 1996).

In the following, a description of the principles behind the model and the governing equations implemented and solved in MIKE SHE are presented. It is implicitly assumed that the equations derived for a hill slope can be applied to describe overland flow, in a lumped manner, across a catchment.

13.2.1 Theoretical basis

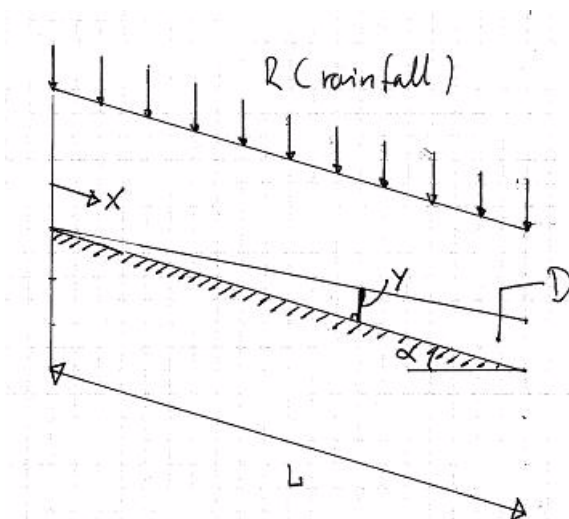


Figure 13.4 Schematic of overland flow on a plane

Figure 13.4 represents a schematic of overland flow on a planar surface of infinite width with uniform rainfall. Precipitation falls on the plane, builds on the surface in response to the surface roughness, and flows down the slope in the positive x-direction. In the figure, L is the length of the slope, y is the local depth of water on the surface at any point along the surface and α is the slope. Then, from the continuity equation

$$\frac{\partial q}{\partial x} = R - \frac{\partial y}{\partial t} \quad (13.20)$$



where q is the specific discharge.

For turbulent flow on a plane of infinite width, the Manning equation is

$$q = M \cdot y^{5/3} \sqrt{\alpha} \quad [m^2/s] \quad (13.21)$$

where M is the Mannings M .

Now, at equilibrium, the depth no longer changes and the specific discharge approaches the rainfall rate

$$\frac{\partial y}{\partial t} = 0 \Rightarrow \frac{\partial q}{\partial x} = R \Rightarrow q_e = R \cdot x \quad (13.22)$$

where q_e is the equilibrium specific discharge.

Then, at equilibrium, the volume of water detained on the surface, D_e , can be calculated by

$$\begin{aligned} D_e &= \int_0^L y dx = \int_0^L \left(\frac{q_e}{M \cdot \sqrt{\alpha}} \right)^{3/5} dx = \int_0^L \left(\frac{R \cdot x}{M \cdot \sqrt{\alpha}} \right)^{3/5} dx \\ D_e &= \frac{5}{8} \cdot \frac{R^{3/5} \cdot L^{8/5}}{M^{3/5} \cdot \alpha^{3/10}} \quad [m^3/m] \end{aligned} \quad (13.23)$$

The depth, y , near the leading edge of the flow plane can be related to the depth at equilibrium by

$$y = \left(\frac{t}{t_e} \right) \cdot y_e \quad (13.24)$$

where t is the time and t_e is the time until the equilibrium is reached.

Then from (13.21) we can write

$$q = M \cdot \left(\frac{t}{t_e} \right)^{5/3} \cdot y_e^{5/3} \sqrt{\alpha} \quad [m^2/s] \quad (13.25)$$

Now if we integrated the specific discharge from time 0 to when the equilibrium is reached, we can calculate the total volume discharged, Q , (per unit width of the plane) by



$$Q = \int_0^{t_e} q dt = \int_0^{t_e} M \cdot \left(\frac{t}{t_e}\right)^{5/3} \cdot y_e^{5/3} \cdot \sqrt{\alpha} (dt)$$

$$Q = \frac{3}{8} \cdot M \cdot y_e^{5/3} \cdot \sqrt{\alpha} \cdot t_e [m^2] \quad (13.26)$$

From (13.25), at equilibrium, ($t = t_e$), the depth of water at the leading edge of the plane ($x = L$) is

$$q = M \cdot y_e^{5/3} \sqrt{\alpha} = R \cdot L \quad (13.27)$$

which yields

$$y_e^{5/3} = \frac{R \cdot L}{M \cdot \sqrt{\alpha}} [m^{5/3}] \quad (13.28)$$

From continuity, the total volume of inflow up until equilibrium must equal the total outflow minus the amount retained on the surface. Thus,

Inflow - Outflow = Surface storage

which from equations (13.26) and (13.23) yields

$$(R \cdot L \cdot t_e) - \left(\frac{3}{8} \cdot M \cdot y_e^{5/3} \cdot \sqrt{\alpha} \cdot t_e\right) = \left(\frac{5}{8} \cdot \frac{R^{3/5} \cdot L^{8/5}}{M^{3/5} \cdot \alpha^{3/10}}\right) \quad (13.29)$$

which, when simplified, yields the time to reach equilibrium

$$t_e = \frac{L^{3/5}}{R^{2/5} \cdot M^{3/5} \cdot \alpha^{3/10}} = \frac{8}{5} \cdot \frac{De}{R \cdot L} \quad (13.30)$$

If we now assume that the flow on the sloping plane is uniform, that is the change in discharge as a function of x is zero, then the depth prior to equilibrium is simply

$$y = R \cdot t \quad (13.31)$$



and the relationship between the depth, y , and the surface storage at equilibrium, D_e , is given by

$$y = \frac{8}{5} \cdot \frac{D_e}{L} [m] \quad (13.32)$$

The relationship between the depth, y , and the detained surface storage prior to equilibrium, D , is given by an empirical model (Fleming, 1975; Crawford and Linsley, 1966)

$$y = \frac{D}{L} \left(1 + 3/5 \cdot \left(\frac{D}{D_e} \right)^3 \right) [m] \quad (13.33)$$

where during the recession part of the hydrograph, when D/D_e is greater than 1, D/D_e is assumed to be equal to 1.

Substituting (13.33) into the Manning equation (13.21) yields

$$q = M \cdot \sqrt{\alpha} \cdot \left[\frac{D}{L} \left(1 + 3/5 \cdot \left(\frac{D}{D_e} \right)^3 \right) \right]^{5/3} [m^2/s] \quad (13.34)$$

13.2.2 Implementation in MIKE SHE

In MIKE SHE the current level of surface detention storage is continually estimated by solving the continuity equation

$$D_2 = D_1 + (\bar{q}_{supply} - \bar{q}) \cdot \Delta t \quad (13.35)$$

where D_1 is the detained storage volume at the start of the time step and D_2 is the detained storage volume at the end of the time step, q is the overland flow during the time interval, and q_{supply} is the amount of water being added to overland flow during the time step. Since q is a function of the average detained storage volume, $(D_1 + D_2)/2$, equation (13.34) is solved iteratively until a solution satisfies both equations.

13.2.3 Coupling to other processes

Overland flow interacts with the other process components, such as evapotranspiration from the water surface, infiltration into the underlying soils, interaction with soil drains, drainage into the channel network, etc. This is an integral part of the MIKE SHE framework and these interactions are treated in the same manner in both the Semi-distributed Overland



Flow Routing model and the 2D Finite Difference Method, based on the diffusive wave approximation.

The Semi-distributed Overland Flow Routing model simulates flow down a hillslope. To apply this at the catchment scale, it is assumed that the overland flow response for a catchment is similar to that of an equivalent hillslope. Furthermore, the drainage of overland flow from one catchment to the next, and from the catchment to the river channels is represented conceptually as a cascade of overland flow reservoirs.

13.2.4 Avoiding the redistribution of ponded water

In the standard version of the Simplified Overland Flow solver, the solver calculates a mean water depth for the entire flow zone using the available overland water from all of the cells in the flow zone. During the Overland flow time step, ET and infiltration are calculated for each cell and lateral flows to and from the zone are calculated. At the end of the time step, a new average water depth is calculated, which is assigned to all cells in the flow zone.

In practice, this results in a redistribution of water from cells with ponded water (e.g. due to high rainfall or low infiltration) to the rest of the flow zone where cells potentially have a higher infiltration capacity. To avoid this redistribution, an option has been added where the solver only calculates overland flow for the cells that can potentially produce runoff, that is, only in the cells for which the water depth exceeds the detention storage depth.

Example application

To illustrate the effect of this option, it was applied to a model with a 10 x 10 square domain, one subcatchment and 3 different soil types in the unsaturated zone with the following saturated hydraulic conductivities

- coarse $1\text{e-}5$ m/s
- medium $1\text{e-}7$ m/s
- fine $1\text{e-}9$ m/s

For rainfall, a synthetic time series with alternating daily values of 50 and 0 mm/day was used. The simulation period was 2 weeks. Thus, the cumulative rainfall input was 350 mm.

For the case, where the ponding water was not redistributed, the cumulative runoff was 96 mm. Whereas, when the ponded water was redistributed, the cumulative runoff was essentially zero.



Activating the option

This option is activated by means of the boolean Extra Parameter, ***Only Simple OL from ponded***, set to **On**. For more information on the use of extra parameters, see Extra Parameters (V.1 p. 145).

13.2.5 Routing simple overland flow directly to the river

In the standard version of the Simplified Overland Flow solver, the water is routed from 'higher' zones to 'lower' zones within a subcatchment. Thus, overland flow generated in the upper zone is routed to the next lowest flow zone based on the integer code values of the two zones. In other words, at the beginning of the time step the overland flow leaving the upper zone (calculated in the previous time step) is distributed evenly across all of the cells in the receiving zone. In practice, this results in a distribution of water from cells in the upstream zone with ponded water (e.g. due to high rainfall or low infiltration) to all of the cells in the downstream zone with potentially a large number of those cells having a higher infiltration capacity. In this case, then, overland flow generated in the upper flow zone may never reach the stream network because it is distributed thinly across the entire downstream zone.

To avoid excess infiltration or evaporation in the downstream zone, an option was added that allows you to route overland flow directly to the stream network. In this case, overland flow generated in any of the overland flow zones is not distributed across the downstream zone, but rather it is added directly to the MIKE 11 stream network as lateral inflow.

Example application

To illustrate the effect of this option, it was applied to a model with a 10 x 10 square domain, one subcatchment and two overland flow zones. The upper zone included an unsaturated zone with a low infiltration capacity, whereas the lower zone had a high infiltration capacity. The saturated hydraulic conductivities of the two zones were

- upper zone 1e-9 m/s
- lower zone 1e-5 m/s

For rainfall, a synthetic time series with alternating daily values of 50 and 0 mm/day was used. The simulation period was 2 weeks. Thus, the cumulative rainfall input was 350 mm.

For the case, where the overland flow was routed directly to the river, the cumulative runoff to the river was 167 mm. Whereas, when the overland flow was routed first to the lower zone, the cumulative runoff reaching the river was only 1 mm.



Activating the option

This option is activated by means of the boolean Extra Parameter, *No Simple OL routing*, set to **On**. For more information on the use of extra parameters, see Extra Parameters (V.1 p. 145).



14 CHANNEL FLOW - REFERENCE

The hydrologic components of MIKE SHE are directly coupled to DHI's river hydraulic program MIKE 11. The MIKE SHE-MIKE 11 coupling enables

- the one-dimensional simulation of river flows and water levels using the fully dynamic Saint Venant equations.
- the simulation of a wide range of hydraulic control structures, such as weirs, gates and culverts.
- area-inundation modelling, using a simple flood-mapping procedure that is based on simulated river water levels and a digital terrain model.
- dynamic overland flooding flow to and from the MIKE 11 river network.
- the full, dynamic coupling of surface and sub-surface flow processes in MIKE 11 and MIKE SHE.

This chapter describes only the interaction between MIKE 11 and MIKE SHE. For technical information on MIKE 11 HD, please refer to either the .pdf version of the MIKE 11 HD Technical Reference Manual that is installed with MIKE 11, or the MIKE 11 documentation in the on-line help.

Surface Water/Aquifer Exchange Mechanisms

In a catchment scale model, it is usually sufficient to consider a river as a line located between model grid cells. In this case, the river-aquifer exchange can be calculated inflow to and from both sides of the river, depending on the head gradient to the adjacent groundwater cells. The line assumption is generally valid if the river width is small relative to the model cells - in other words, catchment or basin scale models.

However, very often a more precise description of the interactions between rivers, flood plains, aquifers and the atmosphere (evapotranspiration) must be adopted. In this context, a reliable description of area-inundation and flood dynamics is crucial.

Thus, the MIKE SHE/MIKE 11 coupling considers three principally different surface water exchange mechanisms:

- **River-Aquifer Exchange (line source/link) (V.2 p. 234)** The river is located on the edge between two adjacent model grid cells. The river is considered a line source/sink to the groundwater and the river is a one-way sink for overland flow.



- **Area Inundation using Flood Codes (areal source/sink) (V.2 p. 239)**
The river has a wide cross-section containing the flood plain and designated cells are “flooded” if the river water level is above the topography.
- **Direct Overbank Spilling to and from MIKE 11 (V.2 p. 240)** The river is a line source/sink, but water above the bank elevation is allowed to flood onto the topography as overland flow.

The above options can be mixed in the MIKE 11 river network, allowing, for example, Flood Codes in the major flood plain and overbank spilling in the upstream secondary branches, but no flooding in the upland regions with steep slopes and narrow channels. MIKE SHE also automatically converts between the line source/sink option and the flooding options. Thus, during low flow conditions, when the river is narrow (less than one grid size) and water flow is confined to the main river channel, the river-aquifer exchange method is adopted. If the river starts to flood one or more model grid cells, MIKE SHE switches to the area-inundation method or floods the grid cells directly via overbank spilling.

14.1 Coupling of MIKE SHE and MIKE 11

The coupling between MIKE 11 and MIKE SHE is made via *river links*, which are located on the edges that separate adjacent grid cells. The river link network is created by MIKE SHE’s set-up program, based on a user-specified sub-set of the MIKE 11 river model, called the *coupling reaches*. The entire river system is always included in the hydraulic model, but MIKE SHE will only exchange water with the coupling reaches. Figure 14.1 shows part of a MIKE SHE model grid with the MIKE SHE river links, the corresponding MIKE 11 coupling reaches, and the MIKE 11 H-points (points where MIKE 11 calculates the water levels).

The location of each of MIKE SHE river link is determined from the coordinates of the MIKE 11 river points, where the river points include both digitised points and H-points on the specified coupling reaches. Since the MIKE SHE river links are located on the edges between grid cells, the details of the MIKE 11 river geometry can be only partly included in MIKE SHE, depending on the MIKE SHE grid size. The more refined the MIKE SHE grid, the more accurately the river network can be reproduced. This also leads to the restriction that each MIKE SHE grid cell can only couple to one coupling reach per river link. Thus, if, for example, the distance between coupling reaches is smaller than half a grid cell, you will probably receive an error, as MIKE SHE tries to couple both coupling reaches to the same river link.



If flooding is not allowed, the MIKE 11 river levels at the H-points are interpolated to the MIKE SHE river links, where the exchange flows from overland flow and the saturated zone are calculated.

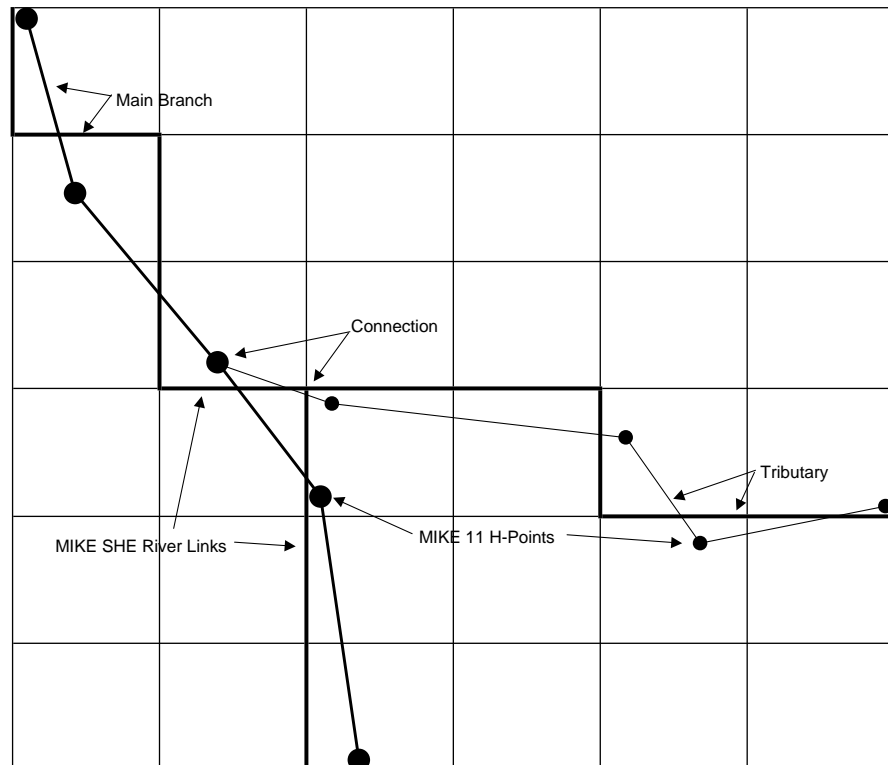


Figure 14.1 MIKE 11 Branches and H-points in a MIKE SHE Grid with River Links

If flooding is allowed, via Flood Codes, then the water levels at the MIKE 11 H-points are interpolated to specified MIKE SHE grid cells to determine if ponded water exists on the cell surface. If ponded water exists, then the unsaturated or saturated exchange flows are calculated based on the ponded water level above the cell.

If flooding is allowed via overbank spilling, then the river water is allowed to spill onto the MIKE SHE model as overland flow.

In each case, the calculated exchange flows are fed back to MIKE 11 as lateral inflow or outflow.



14.1.1 MIKE SHE Branches vs. MIKE 11 Branches

A **MIKE 11 branch** is a continuous river segment defined in MIKE 11. A MIKE 11 branch can be sub-divided into several coupling reaches.

A **MIKE SHE branch** is an unbroken series of coupling reaches of one MIKE 11 branch.

One reason for dividing a MIKE 11 branch into several coupling reaches could be to define different riverbed leakage coefficients for different sections of the river.

If there are gaps between the specified coupling reaches, the sub-division will result in more than one MIKE SHE branch. Gaps of this type are not important to the calculation of the exchange flows between the hydrologic components (e.g. overland to river, or SZ to river). The exchange flows depend on the water level in the MIKE 11 river, which is unaffected by gaps in the coupling reaches.

However, MIKE SHE can calculate how much of the water in the river is from the various hydrologic sources (e.g. fraction from overland flow and SZ exfiltration). However, this sort of calculation is only possible if the MIKE SHE branch is continuous. If there is a gap in a MIKE SHE branch, then the calculated contributions from the different hydrologic sources downstream of the gap will be incorrect. If there are gaps in the MIKE SHE branch network, then the correct contributions from the different sources must be determined from the MIKE 11 output directly.

Furthermore, the MIKE 11/MIKE SHE coupling for the water quality (AD) module will not work correctly if there are gaps in the MIKE SHE branch network.

There is one further limitation in MIKE SHE. That is, no coupling branch can be located entirely within one grid cell. This limitation is to prevent multiple coupling branches being located within a single grid cell.

Connections Between Tributaries and the Main Branch

Likewise, the connections between the tributaries and the main branch are only important for correctly calculating the downstream hydrologic contributions to the river flow and in the advection-dispersion (AD) simulations. The connections are not important to the calculation of the exchange flows between the hydrologic components (e.g. overland to river, or SZ to river).



In the example shown in Figure 14.1, the river links of the tributary are correctly connected to the main branch. This will happen automatically when

- the hydraulic connection is defined in the MIKE 11 network, AND
- the connection point (the chainage) on the main branch is included in a coupling reach, AND
- the connection point (the chainage) on the tributary is included in a coupling reach.

If the connection does not satisfy the above criteria, then there may be a gap in the MIKE SHE branch network and the limitations outlined in the previous section will apply.

14.1.2 The River-Link Cross-section

The MIKE 11(HD) hydraulic model uses the precise cross-sections, as defined in the MIKE 11 *.xns11* (cross-section) file, for calculating the river water levels and the river volumes. However, the exchange of water between MIKE 11 and MIKE SHE is calculated based the river-link cross-section.

The river-link uses is a simplified, triangular cross-section interpolated (distance weighted) from the two nearest MIKE 11 cross-sections. The top width is equal to the distance between the cross-section's left and right bank markers. The elevation of the bottom of the triangle equals the lowest depth of the MIKE 11 cross-section (the elevation of Marker 2 in the cross-section). The top of the river-link equals the elevation of highest bank elevation (left or right bank marker). (See Figure 14.2).

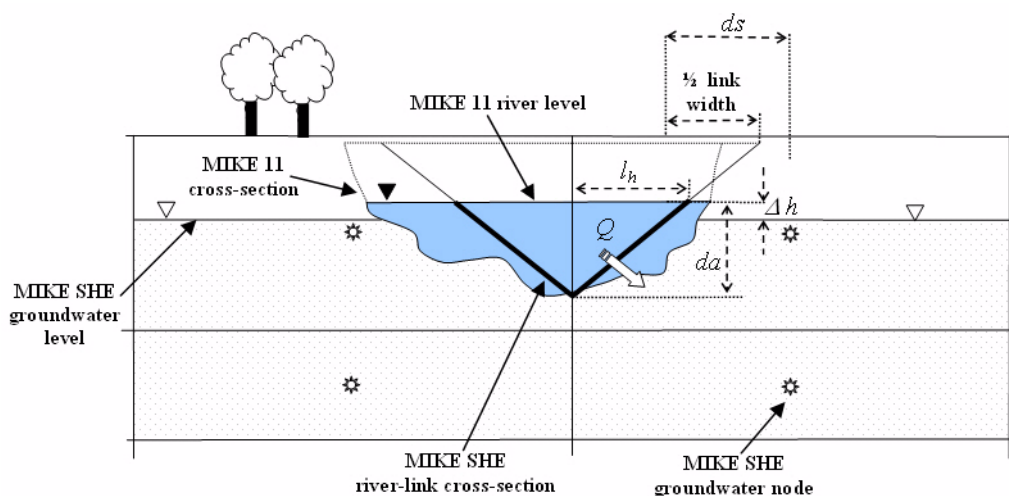


Figure 14.2 A typical simplified MIKE SHE river link cross-section compared to the equivalent MIKE 11 cross-section.

If the MIKE 11 cross-section is wider than the MIKE SHE cell size, then the river-link cross-section is reduced to the cell width. This is a very important limitation, as it embodies the assumption that the river is narrower than the MIKE SHE cell width. If your river is wider than a cell width, and you want to simulate water on the flood plain, then you will need to use either the Area Inundation using Flood Codes (areal source/sink) (V.2 p. 239) option or the Direct Overbank Spilling to and from MIKE 11 (V.2 p. 240) option. If you don't want to simulate flooding, then the reduction of the river link width to the cell width will not likely cause a problem, as MIKE SHE assumes that the primary exchange between the river and the aquifer takes place through the river banks. For more detail on the river aquifer exchange see River-Aquifer Exchange (line source/link) (V.2 p. 234).

14.1.3 Connecting MIKE 11 Water Levels and Flows to MIKE SHE

In MIKE 11, every node in the river network requires information on the river hydraulics, such as cross-section and roughness factors. These nodes are known as H-points, and MIKE 11 calculates the water level at every H-point (node) in the river network. Halfway between each H-point is a Storing Q-point, where MIKE 11 calculates the flow, which must be constant between the H-points.

The water levels at the MIKE 11 H-points are transferred to the MIKE SHE river links using a 2-point interpolation scheme. That is, the water



level in each river link is interpolated from the two nearest H-points, calculated from the centre of the link. The interpolation is proportionally distance-weighted.

The volume of water stored in a river link is based on a sharing of the water in the nearest H-points. In Figure 14.3, River Link A includes all the water volume from H-points 1 and 2, plus part of the volume associated with H-point 3. The volume in River Link B is only related to the volume in H-point 3. While the volume in River Link C includes water from H-points 3 and 4. This is done to ensure consistency between the river volumes in MIKE 11 and MIKE SHE, as the amount of water that can infiltrate is limited by the amount of water stored in the river link.

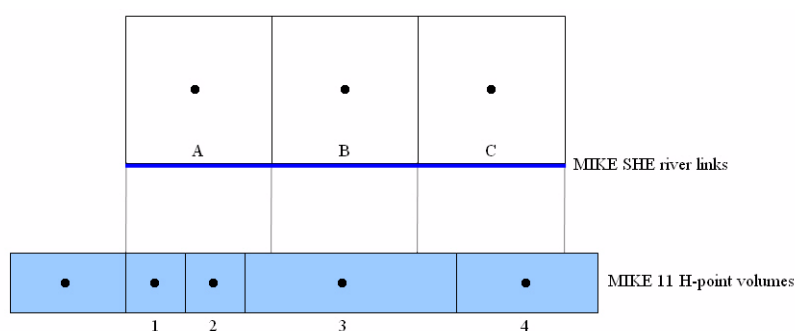


Figure 14.3 Sharing of MIKE 11 H-point volumes with MIKE SHE river links.

The water levels and flows at all MIKE 11 H-points located within the coupling reaches can be retrieved from the MIKE SHE result file. However, since the MIKE 11 flows are not used by MIKE SHE, the river flows stored in the MIKE SHE result file are not the flows calculated at the MIKE 11 Storing Q-points. Rather, the flows stored in the MIKE SHE result file are the estimated flows at the MIKE 11 H-points. That is, the flows in the MIKE SHE result file have been linearly interpolated from the calculated flows at the Storing Q-point locations to the H-point locations on either side of the Storing Q-point. If the exact Q-point discharges are needed, they must be retrieved or plotted directly from the MIKE 11 result file.



14.2 River-Aquifer Exchange (line source/link)

The exchange flow, Q , between a saturated zone grid cell and the river link is calculated as a conductance, C , multiplied by the head difference between the river and the grid cell.

$$Q = C \cdot \Delta h \quad (14.1)$$

Note that Eq. (14.1) is calculated twice - once for each cell on either side of the river link. This allows for different flow to either side of the river if there is a groundwater head gradient across the river, or if the aquifer properties are different.

Referring to Figure 14.2, the head difference between a grid cell and the river is calculated as

$$\Delta h = h_{grid} - h_{riv} \quad (14.2)$$

where h_{grid} is the head in the grid cell and h_{riv} is the head in the river link, as interpolated from the MIKE 11 H-points.

If the ground water level drops below the river bed elevation, the head difference is calculated as

$$\Delta h = z_{bot} - h_{riv} \quad (14.3)$$

where z_{bot} is the bottom of the simplified river link cross section, which is equal to the lowest point in the MIKE 11 cross-section.

In Eq. (14.1), the conductance, C , between the cell and the river link can depend on

- the conductivity of the aquifer material only. See Aquifer Only Conductance (V.2 p. 234), or
- the conductivity of the river bed material only. See River bed only conductance (V.2 p. 235), or
- the conductivity of both the river bed and the aquifer material. See Both aquifer and river bed conductance (V.2 p. 237).

14.2.1 Aquifer Only Conductance

When the river is in full contact with the aquifer material, it is assumed that there is no low permeable lining of the river bed. The only head loss



between the river and the grid node is that created by the flow from the grid node to the river itself. This is typical of gaining streams, or streams that are fast moving.

Thus, referring to Figure 14.2, the conductance, C , between the grid node and the river link is given by

$$C = \frac{K \cdot da \cdot dx}{ds} \quad (14.4)$$

where K is the horizontal hydraulic conductivity in the grid cell, da is the vertical surface available for exchange flow, dx is the grid size used in the SZ component, and ds is the average flow length. The average flow length, ds , is the distance from the grid node to the middle of the river bank in the triangular, river-link cross-section. ds is limited to between 1/2 and 1/4 of a cell width, since the maximum river-link width is one cell width (half cell width per side).

There are three variations for calculating da :

- If the water table is higher than the river water level, da is the saturated aquifer thickness above the bottom of the river bed. Note, however, that da is not limited by the bank elevation of the river cross-section, which means that if the water table in the cell is above the bank of the river, da accounts for overland seepage above the bank of the river.
- If the water table is below the river level, then da is the depth of water in the river.
- If the river cross-section crosses multiple model layers, then da (and therefore C) is limited by the available saturated thickness in each layer. The exchange with each layer is calculated independently, based on the da calculated for each layer. This makes the total exchange independent of the number of layers the river intersects.

This formulation for da assumes that the river-aquifer exchange is primarily via the river banks, which is consistent with the limitation that there is no unsaturated flow calculated beneath the river.

14.2.2 River bed only conductance

If there is a river bed lining, then there will be a head loss across the lining. In this case, the conductance is a function of both the aquifer conductivity and the conductivity of the river bed. However, when the head loss across the river bed is much greater than the head loss in the aquifer material, then the head loss in the aquifer can be ignored (e.g. if the bed mate-



rial is thick and very fine and the aquifer material is coarse). This is the assumption used in many groundwater models, such as MODFLOW.

In this case, referring to Figure 14.2, the conductance, C , between the grid node and the river link is given by

$$C = L_c \cdot w \cdot dx \quad (14.5)$$

where dx is the grid size used in the SZ component, L_c is the leakage coefficient [1/T] of the bed material, and w is the wetted perimeter of the cross-section.

In Eq. (14.5), the wetted perimeter, w , is assumed to be equal to the sum of the vertical and horizontal areas available for exchange flow. From Figure 14.2, this is equal to $da + l_h$, respectively. The horizontal infiltration length, l_h , is calculated based on the depth of water in the river and the geometry of the triangular river-link cross-section.

The infiltration area of the river link closely approximates the infiltration area of natural channels when the river is well connected to the aquifer. In this case, the majority of the groundwater-surface water exchange occurs through the banks of the river and decreases to zero towards the centre of the river. However, in the case of losing streams separated from the groundwater table by an unsaturated zone, the majority of the infiltration occurs vertically and not through the river banks. In this case, the horizontal infiltration area may be too small, if the MIKE 11 bank elevations are much higher than the river level. This can be compensated for by either choosing a lower bank elevation or by increasing the leakage coefficient.

There are three variations for calculating da :

- If the water table is higher than the river water level, da is the saturated aquifer thickness above the bottom of the river bed. Note, however, that da is not limited by the bank elevation of the river cross-section, which means that if the water table in the cell is above the bank of the river, da accounts for overland seepage above the bank of the river.
- If the water table is below the river level, then da is the depth of water in the river.
- If the river cross-section crosses multiple model layers, then da (and therefore C) is limited by the available saturated thickness in each layer. The exchange with each layer is calculated independently, based on the da calculated for each layer. This makes the total exchange independent of the number of layers the river intersects.



This formulation for da assumes that the river-aquifer exchange is primarily via the river banks, which is consistent with the limitation that there is no unsaturated flow calculated beneath the river.

14.2.3 Both aquifer and river bed conductance

If there is a river bed lining, then there will be a head loss across the lining. In this case, the conductance is a function of both the aquifer conductivity and the conductivity of the river bed and can be calculated as a serial connection of the individual conductances. Thus, referring to Figure 14.2, the conductance, C , between the grid node and the river link is given by

$$C = \frac{1}{\frac{ds}{K \cdot da \cdot dx} + \frac{1}{L_c \cdot w \cdot dx}} \quad (14.6)$$

where K is the horizontal hydraulic conductivity in the grid cell, da is the vertical surface available for exchange flow, dx is the grid size used in the SZ component, ds is the average flow length, L_c is the leakage coefficient [1/T] of the bed material, and w is the wetted perimeter of the cross-section. The average flow length, ds , is the distance from the grid node to the middle of the river bank in the triangular, river-link cross-section. ds is limited to between 1/2 and 1/4 of a cell width, since the maximum river-link width is one cell width (half cell width per side).

In Eq. (14.5), the wetted perimeter, w , is assumed to be equal to the sum of the vertical and horizontal areas available for exchange flow. From Figure 14.2, this is equal to $da + l_h$, respectively. The horizontal infiltration length, l_h , is calculated based on the depth of water in the river and the geometry of the triangular river-link cross-section.

The infiltration area of the river link closely approximates the infiltration area of natural channels when the river is well connected to the aquifer. In this case, the majority of the groundwater-surface water exchange occurs through the banks of the river and decreases to zero towards the centre of the river. However, in the case of losing streams separated from the groundwater table by an unsaturated zone, the majority of the infiltration occurs vertically and not through the river banks. In this case, the horizontal infiltration area may be too small, if the MIKE 11 bank elevations are much higher than the river level. This can be compensated for by either choosing a lower bank elevation or by increasing the leakage coefficient.

There are three variations for calculating da :



- If the water table is higher than the river water level, da is the saturated aquifer thickness above the bottom of the river bed. Note, however, that da is not limited by the bank elevation of the river cross-section, which means that if the water table in the cell is above the bank of the river, da accounts for overland seepage above the bank of the river.
- If the water table is below the river level, then da is the depth of water in the river.
- If the river cross-section crosses multiple model layers, then da (and therefore C) is limited by the available saturated thickness in each layer. The exchange with each layer is calculated independently, based on the da calculated for each layer. This makes the total exchange independent of the number of layers the river intersects.

This formulation for da assumes that the river-aquifer exchange is primarily via the river banks, which is consistent with the limitation that there is no unsaturated flow calculated beneath the river.

14.2.4 Steady-state groundwater simulations

For steady-state groundwater models, MIKE 11 is not actually run. Rather the initial water level in MIKE 11 is used for calculating da in the conductance formulas and h_{riv} for the head gradient.

To improve numerical stability during steady-state groundwater simulations, the actual conductance used in the current iteration is an average of the currently calculated conductance and the conductance used in the previous iteration.

Canyon option for steady-state groundwater simulations

In the case of a deep, narrow channel crossing multiple model layers, the head difference used in Equations (14.1) and (14.2) can optionally be limited by the bottom elevation of the layer. Thus,

$$\Delta h = h_{grid} - \max(h_{riv}, z) \quad (14.7)$$

where z is the bottom of the current layer.

The above formulation reduces the infiltration from upper layers by reducing the available gradient. Without the 'Canyon' option, MIKE SHE effectively assumes that the river is hydraulically connected to the upper most model layer, since MIKE SHE calculates the exchange flow with all layers that intersect the river based on the difference between the river level and the water table.



Currently, this option is only available for steady-state models. It is activated by means of the boolean Extra Parameter, *Enable Canyon Exchange*. For more information on the use of extra parameters, see Extra Parameters (V.1 p. 145).

14.3 Area Inundation using Flood Codes (areal source/sink)

The MIKE SHE/MIKE 11 coupling allows you to simulate large water bodies such as lakes and reservoirs, as well as flooded areas. If this option is used, MIKE SHE/MIKE 11 applies a simple flood-mapping procedure where MIKE SHE grid points (e.g. grid points in a lake or on a flood plain) are linked to the nearest H-point in MIKE 11 (where the water levels are calculated). Surface water stages are then calculated in MIKE SHE by comparing the water levels in the H-points with the surface topographic elevations.

14.3.1 Determination of the Flooded Area and Water Levels

The flooded area in MIKE SHE must be delineated by means of integer flood codes, where each coupling reach is assigned a flood code.

During the simulation, the flood-mapping procedure calculates the surface water level on top of each MIKE SHE cell with a flood code by comparing the MIKE 11 surface water level to the surface topography in the model grid. A grid cell is flooded when the MIKE 11 surface water level is above the topography. The MIKE 11 water level is then used as the level of ponded surface water.

The actual water level in the grid cell is calculated as a distance weighted average of the upstream and downstream MIKE 11 H-points.

14.3.2 Calculation of the Exchange Flows

After the MIKE SHE overland water levels have been updated, MIKE SHE calculates the infiltration to the unsaturated and saturated zones and evapotranspiration. Thus, MIKE SHE simply considers any water on the surface, including MIKE 11 flood water as 'ponded water', disregarding the water source. In other words, ponded rainfall and ponded flood water are indistinguishable.

MIKE SHE does not calculate overland flow between cells that are flooded by MIKE 11. Nor, does MIKE SHE calculate overland exchange to MIKE 11, if the cell is flooded by MIKE 11. However, lateral overland flow to neighbouring non-flooded cells is allowed. Thus, if there is a neighbouring, non-flooded cell with a topography lower than a flooded



cell's water level, then MIKE SHE will calculate overland flow to the non-flooded cell as normal.

The calculated exchange flow between the flooded grid cells and the overland, saturated, unsaturated zone or other source/sink terms is fed back to MIKE 11 as lateral inflow or outflow to the corresponding H-point in the next MIKE 11 time step.

In terms of the water balance, the surface water in the inundated areas belongs to the MIKE 11 water balance. In other words, if there is ponded water on the surface when the grid cell floods, the existing ponded water is added to the MIKE 11 water flow in the river. As long as the element is flooded, any exchange to or from the surface water is managed by MIKE 11 as lateral inflow and regular overland flow is not calculated.

If the element reverts back to a non-flooded state, then any subsequent ponded water is again treated as regular overland flow and the water balance is accounted for within the overland flow component.

14.4 Direct Overbank Spilling to and from MIKE 11

If you want to calculate 2D overland flow on the flood plain during a storm event, then you cannot use the Area Inundation using Flood Codes (areal source/sink) (V.2 p. 239) method. The Area Inundation method is primarily used as a way to spread river water onto the flood plain and make it available for interaction with the subsurface via infiltration and evapotranspiration.

The Overbank spilling option treats the river bank as a weir. When the overland flow water level or the river water level is above the left or right bank elevation, then water will spill across the bank based on the standard weir formula

$$Q = \Delta x \cdot C \cdot (H_{us} - H_w)^k \cdot \left[1 - \left(\frac{H_{ds} - H_w}{H_{us} - H_w} \right)^k \right]^{0.385} \quad (14.8)$$

where Q is the flow across the weir, Δx is the cell width, C is the weir coefficient, H_{us} and H_{ds} refer to the height of water on the upstream side and downstream side of the weir respectively, H_w is the height of the weir, and k is a head exponent.

If the water levels are such that water is flowing to the river, then the overland flow to the river is added to MIKE 11 as lateral inflow. If the water



level in the river is higher than the level of ponded water, then the river water will spill onto the MIKE SHE cell and become part of the overland flow.

If the upstream water depth over the weir approaches zero, the flow over the weir becomes undefined. Therefore, the calculated flow is reduced to zero linearly when the upstream height goes below a threshold.

If you use the overbank spilling option, then you should also use the Explicit Numerical Solution (*V.2 p. 217*) for overland flow.





15 **EVAPOTRANSPIRATION - REFERENCE**

The calculation of evapotranspiration uses meteorological and vegetative data to predict the total evapotranspiration and net rainfall due to

- Interception of rainfall by the canopy,
- Drainage from the canopy to the soil surface,
- Evaporation from the canopy surface,
- Evaporation from the soil surface, and
- Uptake of water by plant roots and its transpiration, based on soil moisture in the unsaturated root zone.

In MIKE SHE, the ET processes are split up and modelled in the following order:

- 1 A proportion of the rainfall is intercepted by the vegetation canopy, from which part of the water evaporates.
- 2 The remaining water reaches the soil surface, producing either surface water runoff or percolating to the unsaturated zone.
- 3 Part of the infiltrating water is evaporated from the upper part of the root zone or transpired by the plant roots.
- 4 The remainder of the infiltrating water recharges the groundwater in the saturated zone.

The primary ET model is based on empirically derived equations that follow the work of Kristensen and Jensen (1975), which was carried out at the Royal Veterinary and Agricultural University (KVL) in Denmark.

In addition to the Kristensen and Jensen model, MIKE SHE also includes a simplified ET model that is used in the Two-Layer UZ/ET model. The Two-Layer UZ/ET model divides the unsaturated zone into a root zone, from which ET can occur and a zone below the root zone, where ET does not occur. The Two-Layer UZ/ET module is based on a formulation presented in Yan and Smith (1994). Its main purpose is to provide an estimate of the actual evapotranspiration and the amount of water that recharges the saturated zone. It is primarily suited for areas where the water table is shallow, such as in wetland areas.

Leaf Area Index (LAI)

The area of leaves above a unit area of the ground surface is defined by the leaf area index, *LAI*. Usually, generalised time varying functions of the *LAI* for different crops have been established. Thus, in MIKE SHE, the



user must specify the temporal variation of the LAI for each crop type during the growing seasons to be simulated. Different climatic conditions from year to year may require a shift of the LAI curves in time but will generally not change the shape of the curve. Typically, the LAI varies between 0 and 7.

Root Depth

The root depth is defined as the maximum depth of active roots in the root zone.

15.1 *Kristensen and Jensen method*

The primary ET model is based on empirically derived equations that follow the work of Kristensen and Jensen (1975), which was carried out at the Royal Veterinary and Agricultural University (KVL) in Denmark. In this model, the actual evapotranspiration and the actual soil moisture status in the root zone is calculated from the potential evaporation rate, along with maximum root depth and leaf area index for the plants. The empirical equations in the model are based on actual measurements. The model generally assumes the temperature to be above 0°C and hence, that precipitation does not occur as snow.

In this section, the theory and principles behind the Kristensen and Jensen (1975) evapotranspiration model are presented in detail.

15.1.1 *Sublimation from Snow*

If the air temperature is below the Threshold melting temperature, then the water will be removed from the snow storage as sublimation before any other ET is removed using

$$E_{snow} = Reference_ET * \Delta t \quad (15.1)$$

where Reference_ET refers to the Reference Evapotranspiration (V.2 p. 79) before being reduced by the Crop Coefficient, k_c , that is specified in the Vegetation Development Table (V.2 p. 180). If there is not enough snow storage then E_{snow} will reduce the snow storage to zero.

15.1.2 *Canopy Interception*

Interception is defined as the process whereby precipitation is retained on the leaves, branches, and stems of vegetation. This intercepted water evaporates directly without adding to the moisture storage in the soil.



The interception process is modelled as an interception storage, which must be filled before stem flow to the ground surface takes place. The size of the interception storage capacity, I_{max} , depends on the vegetation type and its stage of development, which is characterised by the leaf area index, LAI . Thus,

$$I_{max} = C_{int} \cdot LAI \quad (15.2)$$

where C_{int} is an interception coefficient [L] and LAI is leaf area index [-].

The coefficient C_{int} defines the interception storage capacity of the vegetation. A typical value is about 0.05 mm but a more exact value may be determined through calibration.

Note The interception coefficient is a unit of length [mm] - not a rate. This means that the full amount is intercepted in every time step, if precipitation is available and the storage is not full. Thus, the total amount of intercepted water is time step dependent. For example, if you have a precipitation rate of 2 mm/hour over 12 hours, the total precipitation will be 24 mm. However, the total interception could range between 2 mm if the time step length is 12 hours to the full 24 mm, if the time step length is 1 hour, assuming that there is 2 mm of evapotranspiration per time step.

15.1.3 Evaporation from the Canopy

The evaporation from the canopy storage is equal to the potential evapotranspiration, if sufficient water has been intercepted on the leaves, that is

$$E_{can} = \min(I_{max}, E_p \Delta t)$$

where E_{can} is the canopy evaporation [LT^{-1}], E_p is the potential evapotranspiration rate [LT^{-1}] and Δt is the time step length for the simulation.

Note: The amount of evaporation from the canopy is time dependent, since the interception on the canopy is calculated for every time step. So, if you half the time step, then the total amount of water stored in the canopy will double. The total amount of water stored in the canopy in temperate climates is generally small compared to the precipitation. However, semi-arid climates, you may impact your water balance.



15.1.4 Plant Transpiration

The transpiration from the vegetation, E_{at} , depends on the density of the crop green material, (i.e. the leaf area index, LAI) the soil moisture content in the root zone and the root density. Thus,

$$E_{at} = f_1(LAI) \cdot f_2(\theta) \cdot RDF \cdot E_p \quad (15.3)$$

where E_{at} is the actual transpiration [LT^{-1}], $f_1(LAI)$ is a function based on the leaf area index [-], $f_2(\theta)$ is a function based on the soil moisture content in the root zone [-], and RDF is a root distribution function [-].

$f_1(LAI)$

The function, $f_1(LAI)$, expresses the dependency of the transpiration on the leaf area of the plant by (see Figure 15.1)

$$f_1(LAI) = C_2 + C_1 LAI \quad (15.4)$$

where C_1 and C_2 are empirical parameters [-].

$f_2(\theta)$

The second function, $f_2(\theta)$, is given by

$$f_2(\theta) = 1 - \left(\frac{\theta_{FC} - \theta}{\theta_{FC} - \theta_w} \right)^{\frac{C_3}{E_p}} \quad (15.5)$$

where θ_{FC} is the volumetric moisture content at field capacity [-], θ_w is the volumetric moisture content at the wilting point [-], θ is the actual volumetric moisture content [-] and C_3 is an empirical parameter [LT^{-1}].

As illustrated in Figure 15.2, higher values of C_3 will lead to higher values of transpiration, which means that the soil will dry out faster, assuming all



other factors constant. In a simulation, the actual transpiration will decrease more quickly for larger values of C_3 .

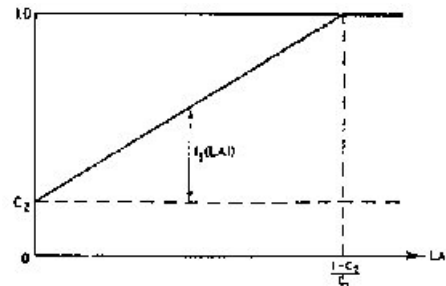


Figure 15.1 The function f_1 versus LAI .

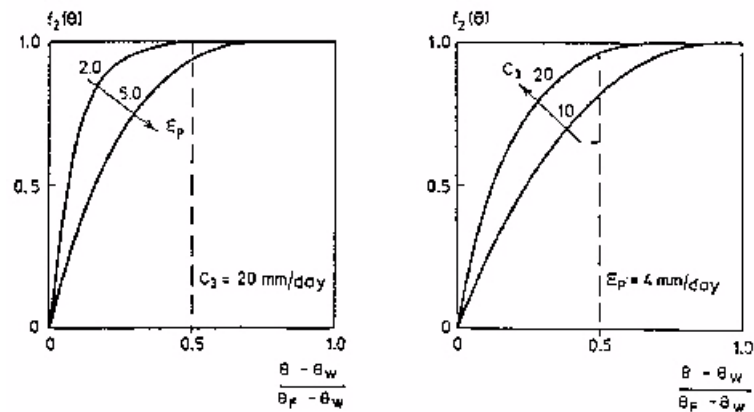


Figure 15.2 The soil moisture function $f_2(\theta)$ for constant C_3 (20 mm/day) and varying E_p (left), and for constant E_p (4 mm/day) and varying C_3 (right).

Root Distribution Function, RDF

Water extraction by the roots for transpiration varies over the growing season. In nature, the exact root development is a complex process, which depends on the climatic conditions and the moisture conditions in the soil.

Thus, MIKE SHE allows for a user-defined, time-varying root distribution determined by the root depth (time varying) and a general, vertical root-density distribution, see Figure 15.3.

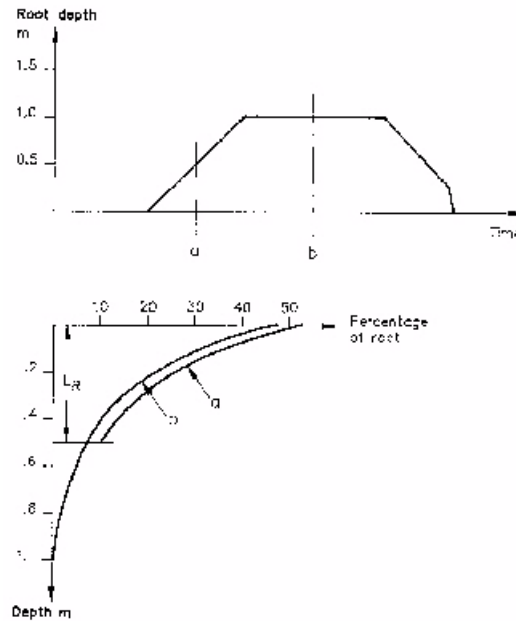


Figure 15.3 Root distribution in time and with depth.

The root extraction is assumed to vary logarithmically with depth by (see Figure 15.3)

$$\log R(z) = \log R_o - AROOT \cdot z \quad (15.6)$$

where R_o is the root extraction at the soil surface, $AROOT$ is a parameter that describes the root mass distribution [-], and z is the depth below ground surface [L].

The value of the Root Distribution Function, RDF, in each layer is then calculated by dividing the amount of water extracted in the layer by the total amount of water extracted by the roots. Thus,

$$RDF_i = \int_{z_1}^{z_2} R(z) dz / \int_0^{L_R} R(z) dz \quad (15.7)$$

where the numerator is the total amount of water extracted in layer i bounded above by Z_1 and below by Z_2 and the denominator is the total



amount of water extracted by the roots between the ground surface and the maximum root depth, L_R .

AROOT

How the water extraction is distributed with depth depends on the *AROOT* parameter. Figure 15.4 shows the distribution of transpiration for different values of *AROOT*, assuming that the transpiration is at the potential rate with no interception loss ($C_{int}=0$) and no soil evaporation loss ($C_2=0$). The figure shows that the root distribution, and the subsequent transpiration, becomes more uniformly distributed as *AROOT* approaches 0. During simulations, the total actual transpiration tends to become smaller for higher values of *AROOT* because most of the water is drawn from the upper layer, which subsequently dries out faster. The actual transpiration, therefore, becomes more dependent on the ability of the soil to conduct water upwards (capillary rise) to the layers with high root density.

Figure 15.5 shows the effect of the root depth, given the same value of *AROOT*. A shallower root depth will lead to more transpiration from the upper unsaturated zone layers because a larger proportion of the roots will be located in the upper part of the profile. However, again, this may lead to smaller actual transpiration, if the ability of the soil to conduct water upwards is limited.

Thus, the factors *AROOT* and root depth are important parameters for estimating how much water can be drawn from the soil profile under dry conditions.

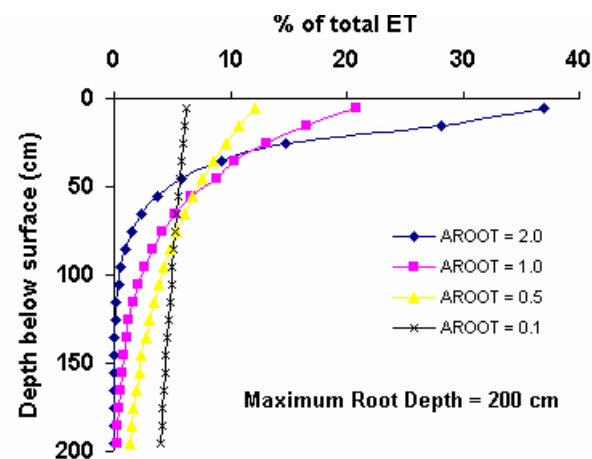


Figure 15.4 Fraction of ET extracted as a function of depth for different values of *AROOT*.

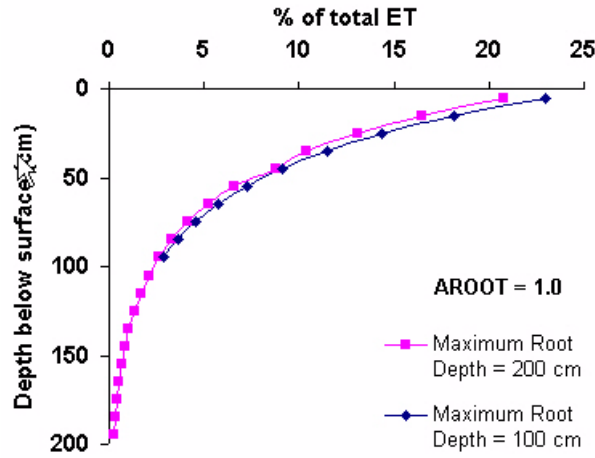


Figure 15.5 Fraction of ET extracted as a function of depth for different maximum root depths.

15.1.5 Soil Evaporation

Soil evaporation, E_s , occurs from the upper part of the unsaturated zone and consists of a basic amount of evaporation, $E_p \cdot f_3(\theta)$, plus additional evaporation from excess soil water as the soil saturation reaches field capacity. This can be described by the following function:

$$E_s = E_p \cdot f_3(\theta) + (E_p - E_{at} - E_p \cdot f_3(\theta)) \cdot f_4(\theta) \cdot (1 - f_1(LAI)) \quad (15.8)$$

where E_p is the potential evapotranspiration, E_{at} is the actual transpiration (Eq. (15.3)), $f_1(LAI)$ is from Eq. (15.4) and the functions $f_3(\theta)$ and $f_4(\theta)$ are given by

$$f_3(\theta) = \begin{cases} C_2 & \text{for } \theta \geq \theta_w \\ C_2 \frac{\theta}{\theta_w} & \text{for } \theta_r \leq \theta \leq \theta_w \\ 0 & \text{for } \theta \leq \theta_r \end{cases}$$

$$f_4(\theta) = \begin{cases} \frac{\theta - \frac{\theta_w + \theta_{FC}}{2}}{\theta_{FC} - \frac{\theta_w + \theta_{FC}}{2}} & \text{for } \theta \geq \frac{(\theta_w + \theta_F)}{2} \\ 0 & \text{for } \theta < \frac{(\theta_w + \theta_F)}{2} \end{cases} \quad (15.9)$$



In the absence of vegetation $f_l(LAI)$ can be set to zero and E_{at} in Eq. (15.8) goes to zero. This allows us to see how E_s varies in relation to E_p for different values of θ . Thus, Eq. (15.8) can be simplified to

$$\frac{E_s}{E_p} = f_3(\theta) + f_4(\theta) - f_3(\theta) \cdot f_4(\theta) \quad (15.10)$$

which is plotted in Figure 15.6.

In the MIKE SHE, soil evaporation is restricted to the upper node in the unsaturated zone, which, generally, should be about 10 centimetres deep, or less.

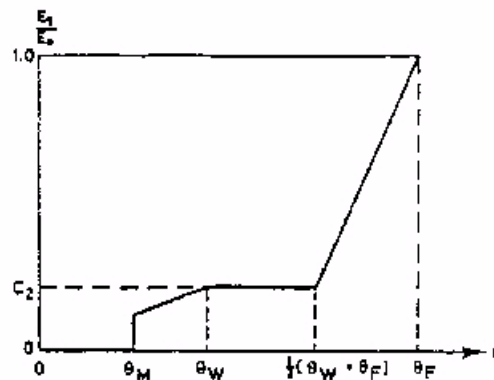


Figure 15.6 Soil evaporation E_s in relation to E_p as a function of θ in the top layer when $f(LAI) = 0$.

15.1.6 Evapotranspiration Coefficients C_1 , C_2 and C_3

The equations for actual transpiration, Eq. (15.3), and soil evaporation, Eq. (15.8), contain three empirical coefficients, C_1 , C_2 , and C_3 . The coefficients C_1 and C_2 are used in the transpiration function, $f_l(LAI)$ (Eq. (15.4)). C_3 is also part of Eq. (15.3), but is only found in the soil moisture function, Eq. (15.5).

C_1

C_1 is plant dependent. For agricultural crops and grass, C_1 has been estimated to be about 0.3. C_1 influences the ratio soil evaporation to transpiration. This is illustrated in Figure 15.7. For smaller C_1 values the soil evaporation becomes larger relative to transpiration. For higher C_1 values,



the ratio approaches the basic ratio determined by C_2 and the input value of LAI.

C_2

For agricultural crops and grass, grown on clayey loamy soils, C_2 has been estimated to be about 0.2. Similar to C_1 , C_2 influences the distribution between soil evaporation and transpiration, as shown in Figure 15.8. For higher values of C_2 , a larger percentage of the actual ET will be soil evaporation. Since soil evaporation only occurs from the upper most node (closest to the ground surface) in the UZ soil profile, water extraction from the top node is weighted higher. This is illustrated in Figure 15.8, where 23 per cent and 61 per cent of the total extraction takes place in the top node for C_2 values of 0 and 0.5 respectively.

Thus, changing C_2 will influence the ratio of soil evaporation to transpiration, which in turn will influence the total actual evapotranspiration possible under dry conditions. Higher values of C_2 will lead to smaller values of total actual evapotranspiration because more water will be extracted from the top node, which subsequently dries out faster. Therefore, the total actual evapotranspiration will become sensitive to the ability of the soil to draw water upwards via capillary action.

C_3

C_3 has not been evaluated experimentally. Typically, a value for C_3 of 20 mm/day is used, which is somewhat higher than the value of 10 mm/day proposed by Kristensen and Jensen (1975). C_3 may depend on soil type and root density. The more water released at low matrix potential and the greater the root density, the higher should the value of C_3 be. Further discussion is given in Kristensen and Jensen (1975).

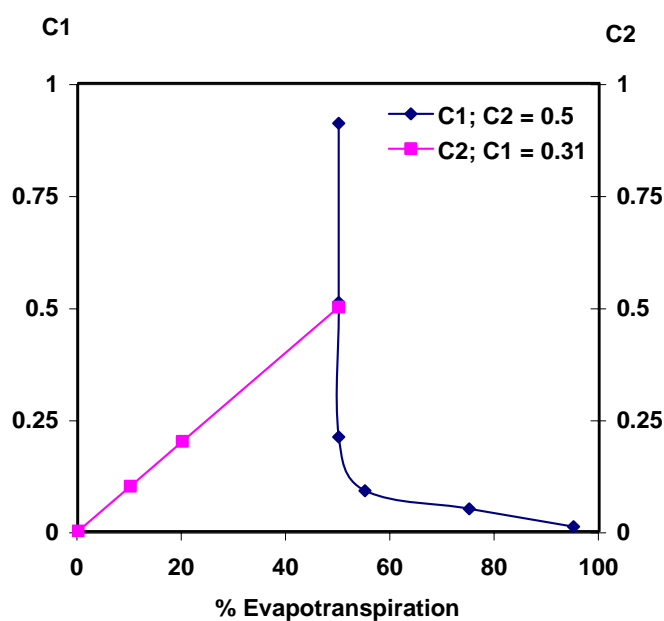


Figure 15.7 The influence of the C_1 and C_2 on the ratio between soil evaporation and transpiration. The values were obtained from model runs assuming $C_{int} = 0$, the moisture content above field capacity, and $LAI = 5$

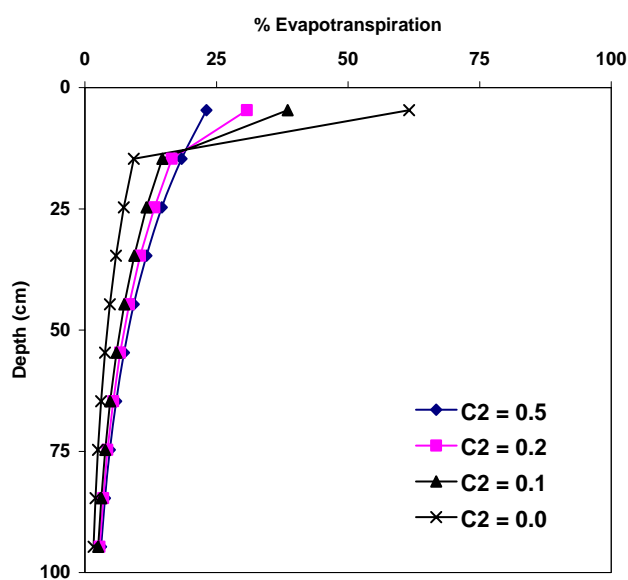


Figure 15.8 Distribution of actual evapotranspiration (in per cent) over depth for different values of C_2 . $C_2=0$ corresponds to pure transpiration.



15.2 Simplified ET for the Two-Layer Water Balance Method

The Two-Layer Water Balance Method is an alternative to the more complex unsaturated flow process coupled to the Kristensen and Jensen module for describing evapotranspiration. The Simplified ET for the Two-Layer Water Balance Method is based on a formulation presented in Yan and Smith (1994). The main purpose of the module is to calculate actual evapotranspiration and the amount of water that recharges the saturated zone.

The module is particularly useful for areas with a shallow ground water table, such as swamps or wetlands areas, where the actual evapotranspiration rate is close to the potential rate. In areas with deeper and drier unsaturated zones, the model does not realistically represent the flow dynamics in the unsaturated zone. The model only considers average conditions and does not account for the relation between unsaturated hydraulic conductivity and soil moisture content and, thereby, the ability of the soil to transport water to the roots. The model simply assumes that if sufficient water is available in the root zone, the water will be available for evapotranspiration. However, it may be possible to “calibrate” the input parameters so that the model performs reasonably well under most conditions.

The Simplified ET module includes the processes of interception, ponding, and evapotranspiration. While MIKE SHE’s unsaturated zone module requires a detailed vertical discretisation of the soil profile (unsaturated zone), the Simplified ET module considers the entire unsaturated zone to be consist of two ‘layers’ representing average conditions in the unsaturated zone.

The input for the model includes the characterisation of the vegetation cover and the physical soil properties. The vegetation is described in terms of leaf area index (LAI) and root depth. The soil properties include a constant infiltration capacity and the soil moisture contents at the wilting point, field capacity and saturation.

The output is an estimate of the actual evapotranspiration and the ground water recharge.



15.2.1 Sublimation from Snow

If the air temperature is below the Threshold melting temperature, then the water will be removed from the snow storage as sublimation before any other ET is removed using

$$E_{snow} = Reference_ET \cdot \Delta t \quad (15.11)$$

where $Reference_ET$ refers to the Reference Evapotranspiration (V.2 p. 79) before being reduced by the Crop Coefficient, k_c , that is specified in the Vegetation Development Table (V.2 p. 180). If there is not enough snow storage then E_{snow} will reduce the snow storage to zero.

15.2.2 Canopy Interception

Interception is defined as the process whereby precipitation is retained on the leaves, branches, and stems of vegetation. This intercepted water evaporates directly without adding to the moisture storage in the soil.

The interception process is modelled as an interception storage, which must be filled before stem flow to the ground surface takes place. The size of the interception storage capacity, I_{max} , depends on the vegetation type and its stage of development, which is characterised by the leaf area index, LAI . Thus,

$$I_{max} = C_{int} \cdot LAI \quad (15.12)$$

where C_{int} is an interception coefficient [mm] and LAI is leaf area index.

The coefficient, C_{int} , defines the interception storage capacity of the vegetation. A typical value is about 0.05 mm but a more exact value may be determined through calibration.

The area of leaves above a unit area of the ground surface is defined by the leaf area index, LAI . Usually, generalised time varying functions of the LAI for different crops have been established. Thus, in MIKE SHE, the user must specify the temporal variation of the LAI for each crop type during the growing seasons to be simulated. Different climatic conditions from year to year may require a shift of the LAI curves in time but will generally not change the shape of the curve. Typically, the LAI varies between 0 and 7.

The actual interception storage, I_a , is then calculated as

$$I_{act} = \min(I_{max}, P \cdot \Delta t)$$



where P is the amount of precipitation and Δt is the calculation time-step.

15.2.3 Soil Moisture

The ET surface, ET_{surf} , is defined as the ground surface less the thickness of the capillary fringe. If the water table is above the ET surface, then ET will not reduce the moisture content of the soil, since any water deficit will be replaced by water drawn up from the saturated zone via capillary action.

The ET extinction depth is the maximum depth to which water can be removed by transpiration. It is defined as the depth of the root zone plus the thickness of the capillary fringe. Thus, if the water table is below the ET extinction depth, then water removed from the root zone by ET cannot be replaced by water drawn up by capillary action, since the roots do not reach the top of the capillary fringe. The depth of the root zone is specified in MIKE SHE's crop database and can vary in time and space.

The simplified ET module assumes that the unsaturated zone can consist of one or two layers. The upper layer extends from the ground surface to the higher of the water table or the ET extinction depth. The second layer extends from the bottom of first layer to the water table, if the water table is below the ET extinction depth. Thus, if the water table is above the ET extinction depth, the thickness of the lower layer is zero. If the water table is at the ground surface then the thickness of the upper layer is also zero. ET is only allowed from the upper of the two ET layers, if the lower layer exists.

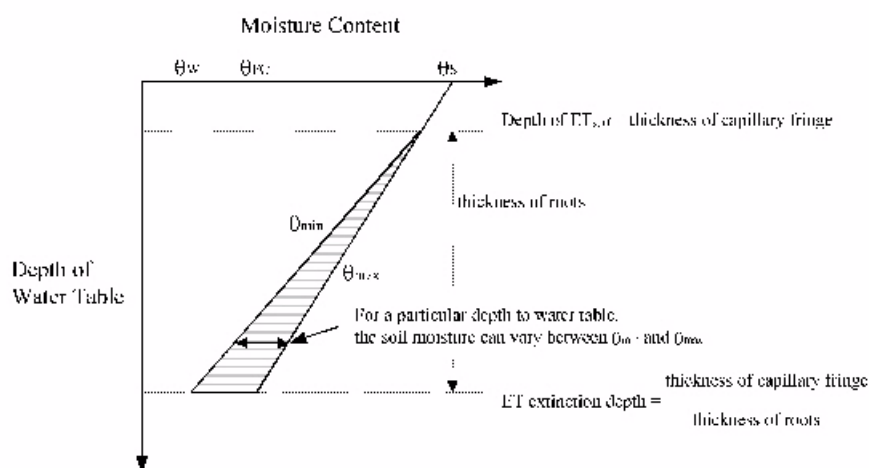


Figure 15.9 Allowable range for soil moisture in the upper ET layer for a given depth to water table.

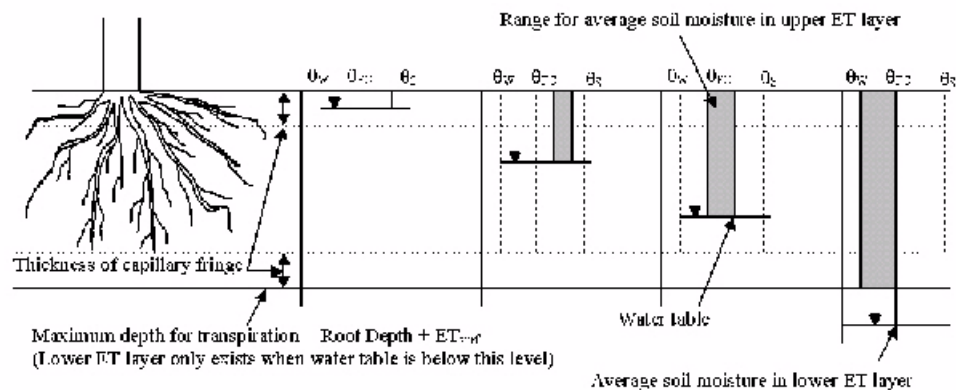


Figure 15.10 Soil moisture ranges for different depths to water table.

If the water table is at the ground surface then the moisture content equals the saturated moisture content, θ_{sat} , and all ET is taken from the saturated zone.

If the water table is below the ground surface, but above the ET surface, then the average moisture content of the ET layer is a linear function of the depth of the water table. That is, the average moisture content in the ET layer is lower when the water table is lower. If the water table is above the ET surface, the capillary fringe reaches the ground surface. Thus, the water content is not dependent on ET and any water lost to ET will be replaced from the groundwater table through capillary action.

If the water table is below the ET surface, but above the ET extinction depth, then the average water content will vary between a minimum, θ_{min} , and a maximum, θ_{max} . θ_{max} is the water content that would be present if no ET occurred. θ_{min} is the minimum water content that can exist in the upper ET layer when ET is active. Both θ_{min} and θ_{max} vary linearly with the depth to the water table. That is, θ_{min} and θ_{max} are lower when the water table is lower.

The difference between θ_{max} and the actual moisture content is the storage capacity of the unsaturated zone. Vertical infiltration to the saturated zone will only occur when the water content is equal to θ_{max} .

If the water table is below the ET extinction depth, then a lower ET layer exists. The moisture content of the lower ET layer is equal to the field capacity, which is the minimum water content when ET does not exist.



The average moisture content of the upper ET layer can range between the field capacity, θ_{FC} , and the wilting point, θ_{WP} , which is the minimum water content at which the plants can remove water from the soil.

15.2.4 Infiltration

At the beginning of each computational time step, rainfall first fills the interception storage. If I_{max} is exceeded, the excess water is added to the amount of ponded water on the ground surface, d_{oc} , which is the height of surface ponding before infiltration is subtracted.

Next, the maximum infiltration volume is limited by the rate of infiltration. Thus,

$$Inf_k = K_{inf} \cdot \Delta t$$

where Inf_k is the maximum amount of infiltration allowed during the time step due to the infiltration rate, K_{inf} is the infiltration rate and Δt is the calculation time-step.

The maximum infiltration volume is also limited by the available storage volume in the unsaturated zone, which is calculated by

$$Inf_v = (\theta_{sat} - \theta_{t-1}) \cdot z_{wt}$$

where θ_{sat} is the saturated water content, θ_{t-1} is the water content at the end of the previous time-step and z_{wt} is the depth of the water table.

The actual infiltration to the unsaturated zone, Inf_{actual} , is then calculated as the minimum of the amount of ponded water before infiltration, the rate limited amount of infiltration or the maximum volume of infiltration. Thus,

$$Inf_{actual} = \min(d_{oc}, Inf_k, Inf_v)$$

Subsequently d_{oc} and θ_{act} are updated

$$d_{oc} = d_{oc}^* - I_{act} \quad [mm]$$

$$\theta_{act} = \theta_{act}^* - (I_{act} / (z_d \cdot 1000)) \quad [-]$$

*where * refers to the parameter value before updating*

15.2.5 Evapotranspiration

Actual evapotranspiration is calculated from the reference evapotranspiration rate (E_p). The reference rate is typically described as a time-series,



which may be derived from pan-measurements or calculated using, for example, the Penman-Monteith equations. The reference ET is satisfied in the following order:

- 1 Evaporation is first deducted from the interception storage assuming the potential ET rate.
- 2 If the interception storage cannot satisfy the potential ET, water is evaporated from the ponded water, d_{oc} , until the ponded water is exhausted or the potential ET is satisfied
- 3 If the potential ET has not yet been satisfied, water is ET is removed from the unsaturated zone until the potential ET is satisfied or the water content of the upper ET layer is reduced to θ_{min} .

If the potential evapotranspiration demand is still not satisfied water is extracted from the saturated zone. The amount that can be extracted is expressed as a function of the depth to the ground water table as described by the MODFLOW ET package.

The actual evapotranspiration is calculated as the sum of the above 4 processes.

ET from the Canopy

Evapotranspiration is deducted from the canopy storage assuming potential evapotranspiration rate. The actual evapotranspiration from canopy, E_{can} is given as minimum of potential evapotranspiration rate multiplied with the time step and actual interception storage:

$$E_{can} = \min(INT_a, E_p \cdot \Delta t) \quad [\text{mm}]$$

INT_a is subsequently updated by deducting E_{can}

$$INT_a = INT_a^* - E_{can} \quad [\text{mm}]$$

ET from Ponded Water

If the interception water storage cannot satisfy potential evapotranspiration rate water is, to the extent possible, removed from the ponded water storage, d_{oc} .

$$E_{pon} = \min(d_{oc}, (E_p - E_{can}) \cdot \Delta t) \quad [\text{mm}] \text{ and } d_{oc} \text{ is updated}$$

$$d_{oc} = d_{oc}^* - E_{pon} \quad [\text{mm}]$$



ET from the Unsaturated Zone

If the potential evapotranspiration demand is still not satisfied water is extracted from the unsaturated zone (if available).

$$E_a = E_a + \min(V_{uz} / dt, E_p - E_a)$$

Where V_{uz} is the available water in the unsaturated zone given as:

$$V_{uz} = (\theta_{act} - \theta_{min}(z_d)) \cdot z_d$$

ET from the Saturated Zone

If the potential evapotranspiration demand is still not satisfied water is extracted from the saturated zone. The amount that can be extracted is expressed as a function of the depth to the ground water table.

$$\begin{aligned} E_{SZ} &= E_p \cdot \Delta t - E_{can} - E_{pon} - E_{UZ} \quad [\text{mm}] \quad z_d < H_c \\ E_{SZ} &= \max\left(E_p \cdot \Delta t - E_{can} - E_{pon} - E_{UZ}, E_p \cdot \Delta t \frac{(h_c + z_{ext} - z_d)}{z_{ext}}\right) \quad H_c \leq z_d < (H_c + z_{ext}) \\ E_{SZ} &= 0 \quad z_d \geq (H_c + z_{ext}) \end{aligned}$$

where

z_{ext} - extinction depth [m]

H_c - ET surface elevation [m]

z_d is considered equal to the root depth. Thus Z_d may be time variant.

Actual ET

Finally, the actual evapotranspiration can be computed as the sum of the above contributions:

$$E_a = E_{can} + E_{pon} + E_{uz} + E_{sz} \quad [\text{mm}]$$

15.2.6 Recharge to the Saturated Zone

If the average water content θ_{act} exceeds the maximum water content (θ_{max}) groundwater recharge (Q_R) is produced.

$$Q_R = \max((\theta_{act} - \theta_{max}(z_d)) \cdot z_d, 0) \quad [\text{mm}]$$



16 **UNSATURATED FLOW - REFERENCE**

Unsaturated flow is one of the central processes in MIKE SHE and in most model applications. The unsaturated zone is usually heterogeneous and characterized by cyclic fluctuations in the soil moisture as water is replenished by rainfall and removed by evapotranspiration and recharge to the groundwater table. Unsaturated flow is primarily vertical since gravity plays the major role during infiltration. Therefore, unsaturated flow in MIKE SHE is calculated only vertically in one-dimension, which is sufficient for most applications. However, this may limit the validity of the flow description in some situations, such as on very steep hill slopes with contrasting soil properties in the soil profile. MIKE SHE includes an iterative coupling procedure between the unsaturated zone and the saturated zone to compute the correct soil moisture and the water table dynamics in the lower part of the soil profile.

There are three options in MIKE SHE for calculating vertical flow in the unsaturated zone:

- the full Richards equation, which requires a tabular or functional relationship for both the moisture-retention curve and the effective conductivity,
- a simplified gravity flow procedure, which assumes a uniform vertical gradient and ignores capillary forces, and
- a simple two-layer water balance method for shallow water tables.

The full Richards equation is the most computationally intensive, but also the most accurate when the unsaturated flow is dynamic. The simplified gravity flow procedure provides a suitable solution when you are primarily interested in the time varying recharge to the groundwater table based on actual precipitation and evapotranspiration and not the dynamics in the unsaturated zone. The simple two-layer water balance method is suitable when the water table is shallow and groundwater recharge is primarily influenced by evapotranspiration in the root zone.

Each cell in the model is assigned to a soil zone, for which a soil profile is defined. In this way, the unsaturated zone can be nominally 'lumped', in so far as the soil profile that is defined for each soil zone represents some sort of average soil profile in the zone. If the depth to the water table is also divided into zones of equal depth, then the unsaturated flow needs only be calculated once for each area with the same soil profile and water table depth. Such lumping can decrease the computational burden considerably. However, when the water table is very dynamic and spatially variable,



there may be no choice but to solve the unsaturated flow equations for each cell in the model using the full Richards solution.

16.1 Richards Equation

The driving force for transport of water in the unsaturated zone is the gradient of the hydraulic head, h , which includes a gravitational component, z , and a pressure component, ψ . Thus,

$$h = z + \psi \quad (16.1)$$

The gravitational head at a point is the elevation of the point above the datum (z is positive upwards). The reference level for the pressure head component is the atmospheric pressure. Under unsaturated conditions the pressure head, ψ , is negative due to capillary forces and short range adsorptive forces between the water molecules and the soil matrix. These forces are responsible for the retention of water in the soil. As these two forces are difficult to separate, they are incorporated into the same term. Although the physical phenomena creating the pressure head under unsaturated and saturated conditions are very different, the pressure head is considered to be a continuous function across the water table, with the pressure being negative above and positive below the water table.

For vertical flow, the driving force for the transport of water is the vertical gradient of the hydraulic head. Thus,

$$\Delta h = \frac{\partial h}{\partial z} \quad (16.2)$$

The volumetric flux is then obtained from Darcy's law:

$$q = -K(\theta) \frac{\partial h}{\partial z} \quad (16.3)$$

where $K(\theta)$ is the unsaturated hydraulic conductivity. Assuming that the soil matrix is incompressible and the soil water has a constant density, the continuity equation will be:

$$\frac{\partial \theta}{\partial t} = -\frac{\partial q}{\partial z} - S(z) \quad (16.4)$$



where θ is the volumetric soil moisture and S is the root extraction sink term. Combining Eqs. (16.1), (16.3) and (16.4) yields

$$\frac{\partial \theta}{\partial t} = \frac{\partial}{\partial z} \left(K(\theta) \frac{\partial \psi}{\partial z} \right) + \frac{\partial K(\theta)}{\partial z} - S(z) \quad (16.5)$$

The dependent variables, θ and ψ , in Eq. (16.5) are related through the hydraulic conductivity function, $K(\theta)$, and the soil moisture retention curve, $\psi(\theta)$

Eq. (16.5) is general, in the sense that it is equally valid in both homogeneous and heterogeneous soil profiles, and there are no constraints on the hydraulic functions.

Introducing the concept of soil water capacity

$$C = \frac{\partial \theta}{\partial \psi} \quad (16.6)$$

which is the slope on the soil moisture retention curve, then the tension-based version of Eq. (16.5) is

$$C \frac{\partial \psi}{\partial t} = \frac{\partial}{\partial z} \left(K(\theta) \frac{\partial \psi}{\partial z} \right) + \frac{\partial K(\theta)}{\partial z} - S \quad (16.7)$$

This equation is usually referred to as Richards equation, which is named after L.A. Richards who first used it in 1931. It still applies when ψ becomes positive, in which case the equation degenerates to the LaPlace equation.

The sink terms in Eq. (16.7) are calculated from the root extraction for the transpiration in the upper part of the unsaturated zone. The integral of the root extraction over the entire root zone depth equals the total actual evapotranspiration. Direct evaporation from the soil is calculated only for the first node below the ground surface.

16.1.1 Numerical Solution

MIKE SHE uses a fully implicit formulation in which the space derivatives of Eq. (16.7) are described by their finite difference analogues at time level $n+1$. The values of $C(\theta)$ and $K(\theta)$ are referred to at time level $n+1/2$. These are evaluated in an iterative procedure averaging C^n , K^n with C^m , K^m respectively. C^m and K^m are calculated as a running average of the coefficients found in each iteration.



This solution technique has been found to eliminate stability and convergence problems arising from the non-linearity of the soil properties.

For an interior node, the implicit scheme yields the following discrete formulation of the vertical flow:

$$q_{J+1/2}^{n+1} = -K_{J+1/2}^{n+1/2} \left(\frac{\psi_{J+1}^{n+1} - \psi_J^{n+1}}{\Delta Z_{J+1}} + 1 \right) \quad (16.8)$$

where the subscript J refers to the spatial increment and the superscript n refers to the time increment. The vertical grid system for a soil column is shown in Figure 16.1. Similar to Eq. (16.8) the discrete form of Eq. (16.1) gives

$$\begin{aligned} C_{J+1}^{n+1} \frac{\psi_J^{n+1} - \psi_J^n}{\Delta t} = & \left[K_{J+1/2}^{n+1/2} \left(\frac{\psi_{J+1}^{n+1} - \psi_J^{n+1}}{\Delta Z_{J+1}} \right) \right. \\ & \left. - K_{J-1/2}^{n+1/2} \left(\frac{\psi_J^{n+1} - \psi_{J-1}^{n+1}}{\Delta Z_J} \right) \right] \frac{1}{1/2(\Delta Z_{J+1} + \Delta Z_J)} \\ & - S_J^{n+1} \end{aligned} \quad (16.9)$$

The soil property K is centred in space using the arithmetic mean:

$$K_{J+1/2}^{n+1/2} = \frac{K_{J+1}^{n+1/2} + K_J^{n+1/2}}{2} \quad K_{J-1/2}^{n+1/2} = \frac{K_J^{n+1/2} + K_{J-1}^{n+1/2}}{2}$$

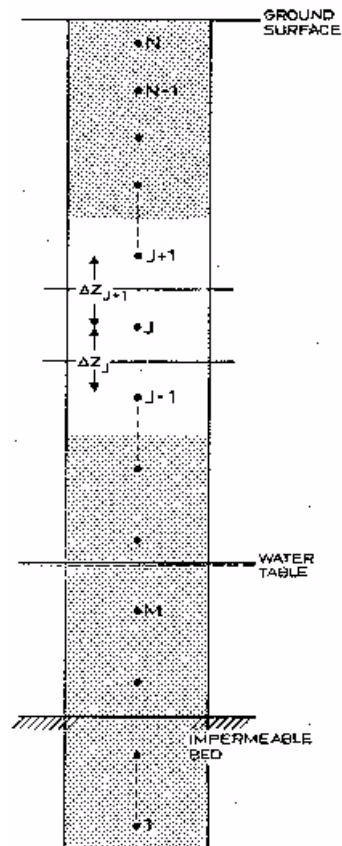


Figure 16.1 Vertical Discretisation in the Unsaturated Zone.

Eq. (16.9) involves three unknown values at time $n+1$ and one known value at time n for each node. Written for all nodes with reference to



Figure 16.1, a system of $N-M+1$ equations with $N-M+1$ unknowns is obtained. The system of equations forms a tri-diagonal matrix:

$$\begin{bmatrix}
 B_N & A_N & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\
 G_{N-1} & B_{N-1} & A_{N-1} & \dots & \dots & \dots & \dots & \dots & \dots \\
 \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\
 \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\
 \dots & \dots & \dots & G_J & B_J & A_J & \dots & \dots & \dots \\
 \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\
 \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\
 \dots & \dots & \dots & \dots & \dots & \dots & G_{M+1} & B_{M+1} & A_{M+1} \\
 \dots & \dots & \dots & \dots & \dots & \dots & \dots & G_M & B_M
 \end{bmatrix}
 \begin{bmatrix}
 \psi_N \\
 \psi_{N-1} \\
 \dots \\
 \dots \\
 \psi_J \\
 \dots \\
 \dots \\
 \psi_{M+1} \\
 \psi_M
 \end{bmatrix}
 =
 \begin{bmatrix}
 D_N \\
 D_{N-1} \\
 \dots \\
 \dots \\
 D_J \\
 \dots \\
 \dots \\
 D_{M+1} \\
 D_M
 \end{bmatrix}
 \quad (16.10)$$

The J 'th row in the matrix is

$$A_J^{n+1} \psi_{J-1} + B_J^{n+1} \psi_J + G_J^{n+1} \psi_{J+1} = D_J \quad (16.11)$$

where

$$\begin{aligned}
 A_J &= -K_{j-\frac{1}{2}}^{n+\frac{1}{2}} / \Delta Z_- \\
 B_J &= C_J^{n+\frac{1}{2}} / \Delta t + K_{j+\frac{1}{2}}^{n+\frac{1}{2}} / \Delta Z_+ + K_{j+\frac{1}{2}}^{n-\frac{1}{2}} / \Delta Z_- \\
 G_J &= -K_{j+\frac{1}{2}}^{n+\frac{1}{2}} / \Delta Z_+ \\
 D_J &= C_J^{n+\frac{1}{2}} \psi_J^n / \Delta t + \frac{K_{j+\frac{1}{2}}^{n+\frac{1}{2}} - K_{j-\frac{1}{2}}^{n+\frac{1}{2}}}{\frac{1}{2}(\Delta Z_{J+1} + \Delta Z_J)} - S_J^{n+1} \\
 \Delta Z_+ &= \frac{1}{2} \Delta Z_{J+1} (\Delta Z_{J+1} + \Delta Z_J) \\
 \Delta Z_- &= \frac{1}{2} \Delta Z_J (\Delta Z_{J+1} + \Delta Z_J)
 \end{aligned} \quad (16.12)$$

The solution to the matrix system Eq. (16.10) is solved by Gaussian elimination. Assuming that ψ_J^{n+1} and ψ_{J+1}^{n+1} can be related in the following recurrence relation

$$\psi_J^{n+1} = E_{J+1} \psi_{J+1}^{n+1} + F_{J+1} \quad (16.13)$$



The E_{J+I} and F_{J+I} can be calculated by combining Eqs. (16.11) and (16.13) as follows:

$$E_{J+1} = \frac{-G_J}{A_J E_J + B_J} \quad F_{J+1} = \frac{D_J - A_J F_J}{A_J E_J + B_J} \quad (16.14)$$

Given the boundary conditions at the bottom and top nodes, ψ is computed for all nodes in a double sweep procedure:

- 1 E and F values is calculated from Eqs. 16.12 and 16.14 for all nodes from bottom-to-top in a E, F -sweep.
- 2 ψ is then calculated from Eq. 16.13 for all nodes in a top-to-bottom sweep.

Briefly, the iterative procedure within each time step is

- 1 the final result at time n (i.e. C_J^n and K_J^n) is used for the initial estimate of C_J^0 and K_J^0 for the first iteration,
- 2 then the following convergence criteria are checked for every node after each iteration, i

$$abs(\psi^i - \psi^{i-1}) < tolerance\ criteria \quad for\ |\psi| < 0.5 \quad (16.15)$$

$$abs\left(\frac{\psi^i - \psi^{i-1}}{\psi^i}\right) < tolerance\ criteria \quad for\ (|\psi| \geq 0.5) \quad (16.16)$$

- 3 if either of these convergence criteria is satisfied then a solution at the current time level (i.e. $n+1$) has been found.
- 4 if the criteria are not fulfilled then C_J^{i+1} and K_J^{i+1} are updated for the next iteration by

$$C_J^{i+1} = 1/2 \left(\sum_{m=1}^i C_J^m / i C_J^0 \right) \quad (16.17)$$

$$K_J^{i+1} = 1/2 \left(\sum_{m=1}^i K_J^m / i K_J^0 \right) \quad (16.18)$$



16.1.2 Boundary Conditions

The unsaturated zone extends from the ground surface to the groundwater table. The vertical flow is determined by the boundary conditions at each end of the column. However, the UZ column only exchanges water with the upper node of the SZ model, even if the UZ model extends below the top layer of the SZ model (see Limitations (V.2 p. 287)).

Upper boundary

The upper boundary condition is either

- a constant flux condition within each time step (Neumann boundary condition), which is determined by the infiltration rate, or
- a constant head condition within each time step (Dirichlet boundary condition), which is determined by the level of ponded water on the surface.

If the infiltration is equal to the net rainfall rate at the soil surface, R , Eq. (16.9) can be written for the top node N as

$$C_N^{n+1/2} \frac{\Psi_N^{n+1} \Psi_N^n}{\Delta t} = \left(-R - K_{N-1/2}^{n+1/2} \left(\frac{\Psi_N^{n+1} - \Psi_{N-1}^{n+1}}{\Delta Z_N} + 1 \right) \right) \frac{1}{0.5(\Delta Z_{N+1} + \Delta Z_N)} - S_N^{n+1} \quad (16.19)$$

where R is defined negative downwards.

Writing Eq. (16.19) in a similar form to Eq. (16.11) yields

$$A_N \Psi_{N-1}^{n+1} + B_N \Psi_N^{n+1} = D_N \quad (16.20)$$

where

$$\begin{aligned} A_N &= -K_{N-1/2}^{n+1/2} / \Delta Z_N \\ B_N &= C_N^{n+1/2} / \Delta t + K_{N-1/2}^{n+1/2} / \Delta Z_N \\ D_N &= C_N^{n+1/2} \frac{\Psi_N^n}{\Delta t} + \frac{-R - K_{N-1/2}^{n+1/2}}{1/2(\Delta Z_{N+1} + \Delta Z_N)} - S_N^{n+1} \end{aligned} \quad (16.21)$$



If water is ponded on the ground surface, the first node is assumed saturated and the boundary condition simply becomes

$$\psi_N^{n+1} = \psi_N^n = \Delta Z_{N+1} \quad (16.22)$$

At the beginning of each UZ time step, the amount of available water for infiltration is calculated as the amount of ponded water, plus the net rainfall at the ground surface, minus evaporation from ponded water.

The upper boundary condition is applied depending on the deficit in the uppermost UZ node:

- If the available water exceeds the deficit in the top UZ node, then the head boundary is used.
- If the available water is less than the deficit in the top UZ node, then a flux boundary is used.

If the head boundary is used, then when the solution is found, the amount of infiltration is compared against the available amount of infiltration. If the available infiltration is exceeded then the solution is repeated with the flux boundary.

If the flux boundary is used, then the available water for infiltration is divided by the time step length to get the infiltration rate. When the solution is found, the water content in the uppermost UZ node is compared to the saturated water content. If the saturated water content was reached or exceeded, then the solution is repeated using the head boundary.

The solution is restricted to a maximum of one repeat in each time step, to prevent an infinite loop.

Lower Boundary

In most cases, the lower boundary is a pressure boundary that is determined by the water table elevation. Then Eq. (16.21) consists of N-M equations. If node M is the first node below the water table, then

$$E_{M+1} = 0 \quad F_{M+1} = \psi_M^{n+1} = h \quad (16.23)$$

where h is the distance between the water table and node M. Noted that ψ_M is independent of ψ_{M+1} since $E_{M+1} = 0$.

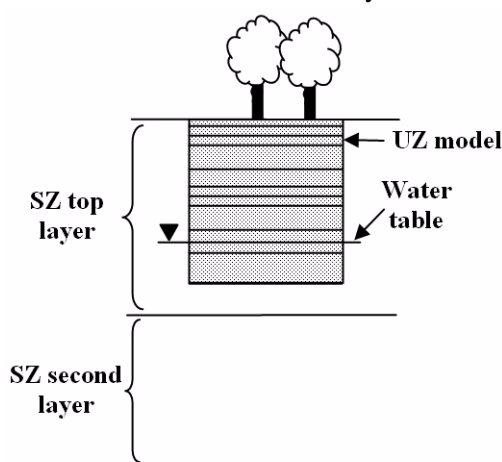
If the UZ model is not coupled to a SZ model, then the lower boundary is automatically converted from a pressure head boundary to a zero flux



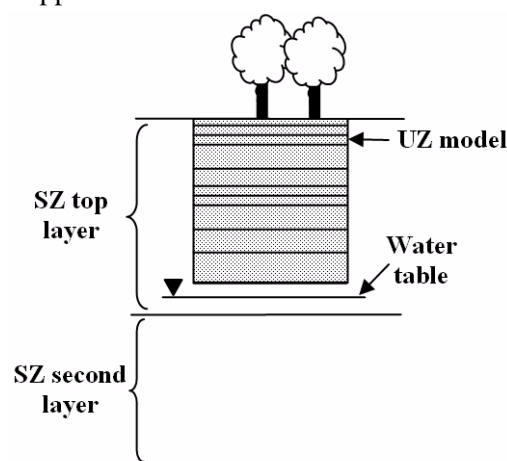
boundary ($Q=0$) if the water table falls below the impermeable bed (see Figure 16.1) and, at same time there is an upward flux in the lower part of the profile. The head boundary is re-started as soon as a positive hydraulic pressure gradient is calculated or the water table starts to rise in the profile.

However, when the UZ model is coupled to an SZ model, the UZ model exchanges water only with the top node of the SZ model. This can lead to three principle conditions:

- If the UZ model intersects the water table in the top layer of the SZ model, then the lower boundary is a normal pressure boundary.

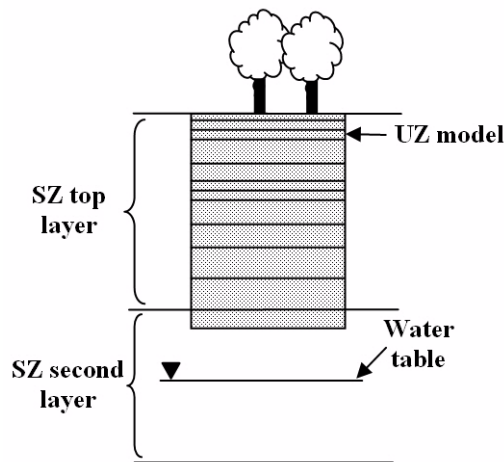


- If the UZ model does not extend to the bottom of the uppermost SZ layer, and the water table in the SZ model falls below the bottom UZ layer, then an error will be generated and the simulation will be stopped.





- If the UZ model extends below the top SZ layer and the top SZ layer dries out, then the UZ model treats the bottom boundary as either a pressure boundary with the pressure equal to the elevation of the bottom of the uppermost SZ layer, or a zero-flux boundary if there is an upward gradient at the lower boundary. The zero-flux boundary is only used for the Full Richards Equation option because the tension term can yield an upwards flow from the groundwater table, which is not physically possible when the upper SZ layer is dry.



In the first and last cases above, the flux out the bottom of the UZ column is added as a flux boundary condition to the uppermost SZ node.

16.1.3 Initial Conditions

The initial conditions for ψ are generated by MIKE SHE assuming an equilibrium soil moisture/pressure profile with no-flow. The equilibrium profile is calculated assuming hydrostatic conditions, as illustrated in Figure 16.2. The pressure decreases linearly from zero at the groundwater table to ψ_{FC} when the moisture content reaches the field capacity and is then remains constant for all nodes above this point. The assumption is that the flow is (almost) zero at moisture contents below the field capacity.

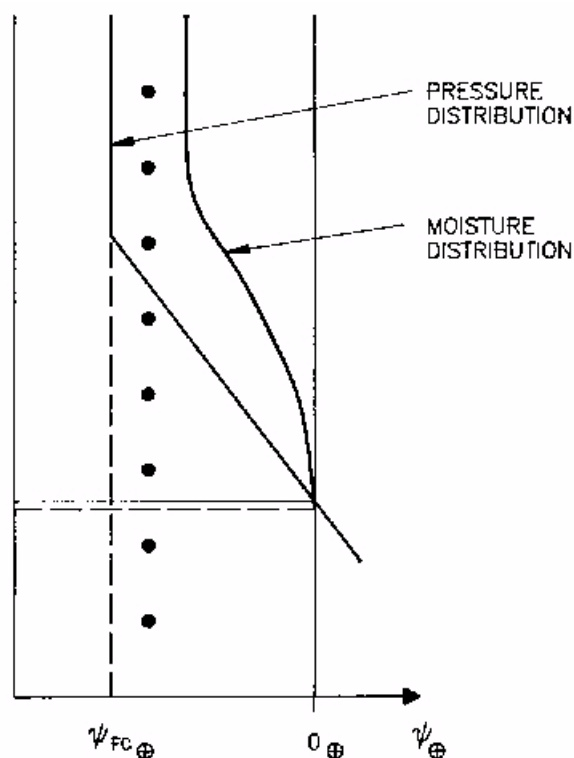


Figure 16.2 Illustration of initial soil moisture profile and pressure head profile.

16.1.4 Sources and sinks

Richards Equation (Eq. (16.7)) includes a source/sink term for each computational node. These sink terms are calculated from the root extraction due to transpiration in the upper part of the unsaturated zone. The integral of the root extraction over the entire root zone depth equals the total actual evapotranspiration. Direct evaporation from the soil is calculated only for the first node in the soil column.

16.1.5 Spatial resolution

The finite difference method assumes that the soil profile is divided into discrete computational nodes, in which the dependent variable is calculated. The non-linearity of the unsaturated flow process creates large gradients in soil pressure and soil moisture content during infiltration. Therefore, it is important to select appropriate nodal increments, so as to describe the flow process with sufficient accuracy but at the same time keeping the computational time reasonable. This trade off can become especially constraining in catchment-scale simulations.



The simulation of Hortonian ponding at the ground surface (high rainfall intensity on dry, low permeable soil) requires a fine spatial resolution in the upper part of the profile (see Figure 16.3). Deeper in the profile the gradients are smaller and larger node increments can usually be selected.

Thus, as a general guideline, one should choose a finer spatial resolution in the top nodes.

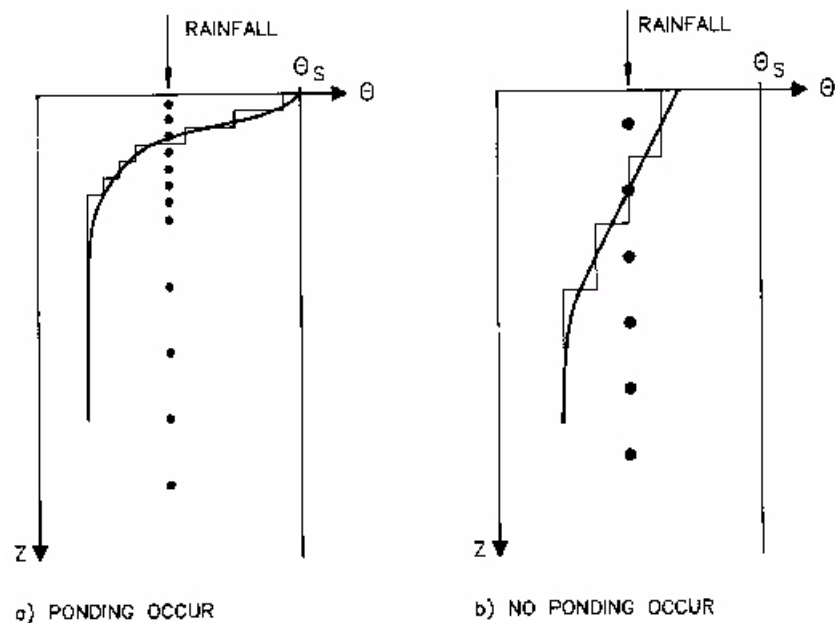


Figure 16.3 Examples of different increments in the soil profile and the resulting water content distribution of 1-3 cm for detailed studies and 3-5 cm for catchment studies. Further down in the profile larger increments can be chosen ranging from 10 cm to 30 cm.

16.2 Gravity Flow

The driving force for transport of water in the unsaturated zone is the gradient of the hydraulic head, h , which includes a gravitational component, z , and a pressure component, ψ . Thus,

$$h = z + \psi \quad (16.24)$$

The gravitational head at a point is the elevation of the point above the datum (z is positive upwards). The reference level for the pressure head component is the atmospheric pressure. Under unsaturated conditions the pressure head, ψ , is negative due to capillary forces and short range



adsorptive forces between the water molecules and the soil matrix. However, in the gravity flow module, the pressure head term is ignored and the driving force is due entirely to gravity.

Thus for vertical flow, the vertical gradient of the hydraulic head becomes,

$$\Delta h = \frac{\partial z}{\partial z} = 1 \quad (16.25)$$

The volumetric flux is then obtained from Darcy's law:

$$q = -K(\theta) \frac{\partial h}{\partial z} = -K(\theta) \quad (16.26)$$

where $K(\theta)$ is the unsaturated hydraulic conductivity. Assuming that the soil matrix is incompressible and the soil water has a constant density, the continuity equation will be:

$$\frac{\partial \theta}{\partial t} = -\frac{\partial q}{\partial z} - S(z) \quad (16.27)$$

where S is the root extraction sink term.

16.2.1 Solution method

In the Gravity Flow Module, Equation (16.27) is solved explicitly from the top of the soil column downward.

At the top of the soil column the infiltration rate is first set equal to the amount of water available for infiltration, which is the depth of overland water on the ground surface. This is reduced to the saturated conductivity of the first unsaturated soil cell, which is the maximum infiltration rate for the soil column (Equation (16.26)).

The infiltration rate is further reduced if a leakage coefficient has been specified for the overland-unsaturated zone interface, which may be done in paved areas or under lakes. A leakage coefficient must be explicitly specified for paved areas that are specified as part of the overland flow routing system. That is, paved areas may be defined as part of the overland flow module to route water to streams from parking lots, etc. However, any reduction in the leakage coefficient under such paved areas must be explicitly defined in the unsaturated zone module. For example, in an model cell where 25% of the land area is paved, a leakage coefficient may



be specified equal to 0.25 times the hydraulic conductivity of the surficial soil.

If the water table is above the ground surface the infiltration is set to zero.

In the special case that the water table is above the top node of the soil column but below the ground surface, the infiltration rate is reduced to an estimate of the moisture deficit in the top cell. This is done to reduce or prevent artificial cycling of water between the unsaturated zone and ponded water on the surface.

If there is sufficient water in the top cell at the start of the time step (water content sufficiently above field capacity to satisfy root extraction), or if there is sufficient net infiltration to raise the moisture content above the field capacity, then the flux through the top cell is calculated based on the hydraulic conductivity, which is a function of the moisture content. The flux is first calculated based on the moisture content at the start of the time step and an updated moisture content is calculated. Then the flux is calculated again based on the updated moisture content and another moisture content is calculated. The actual flux through the cell is then set to the average of these two fluxes. Similarly, the actual updated moisture content is set to the average of the two moisture contents.

This flux is then added the cell below and the calculation repeated downwards for the remaining cells in the column.

Once the water table is reached, the water contents in the cells are rebalanced from the bottom up to ensure that no cell is over saturated.

The flux out the bottom of the soil column is accumulated over the UZ time steps and added as a source to the saturated zone calculation at the start of the next SZ time step.

16.2.2 Initial Conditions

The initial water content for each node in the soil column is calculated based on the moisture retention curves for each of the soil types in the soil column. However, if the calculated pressure head is below the pressure head corresponding to the field capacity of the soil, then the initial moisture content is set to the field capacity.

16.3 Two-Layer Water Balance

The Two-Layer Water Balance Method is an alternative to the more complex unsaturated flow process coupled to the Kristensen and Jensen mod-



ule for describing evapotranspiration. The Simplified ET for the Two-Layer Water Balance Method is based on a formulation presented in Yan and Smith (1994). The main purpose of the module is to calculate actual evapotranspiration and the amount of water that recharges the saturated zone.

The module is particularly useful for areas with a shallow ground water table, such as swamps or wetlands areas, where the actual evapotranspiration rate is close to the potential rate. In areas with deeper and drier unsaturated zones, the model does not realistically represent the flow dynamics in the unsaturated zone. The model only considers average conditions and does not account for the relation between unsaturated hydraulic conductivity and soil moisture content and, thereby, the ability of the soil to transport water to the roots. The model simply assumes that if sufficient water is available in the root zone, the water will be available for evapotranspiration. However, it may be possible to “calibrate” the input parameters so that the model performs reasonably well under most conditions.

The module includes the processes of interception, ponding, infiltration, evapotranspiration and ground water recharge. While MIKE SHE’s unsaturated zone module requires a detailed vertical discretisation of the soil profile (unsaturated zone), the simplified ET module considers the entire unsaturated zone to consist of two ‘layers’ representing average conditions in the unsaturated zone.

The input for the model includes the characterisation of the vegetation cover and the physical soil properties. The vegetation is described in terms of leaf area index (LAI) and root depth. The soil properties include a constant infiltration capacity and the soil moisture contents at the wilting point, field capacity and saturation.

The output is an estimate of the actual evapotranspiration and the ground water recharge.

16.3.1 Soil Moisture

The ET surface, ET_{surf} , is defined as the ground surface less the thickness of the capillary fringe. If the water table is above the ET surface, then ET will not reduce the moisture content of the soil, since any water deficit will be replaced by water drawn up from the saturated zone via capillary action.

The ET extinction depth is the maximum depth to which water can be removed by transpiration. It is defined as the depth of the root zone plus the thickness of the capillary fringe. Thus, if the water table is below the



ET extinction depth, then water removed from the root zone by ET cannot be replaced by water drawn up by capillary action, since the roots do not reach the top of the capillary fringe. The depth of the root zone is specified in MIKE SHE's crop database and can vary in time and space.

The simplified ET module assumes that the unsaturated zone can consist of one or two layers. The upper layer extends from the ground surface to the higher of the water table or the ET extinction depth. The second layer extends from the bottom of first layer to the water table, if the water table is below the ET extinction depth. Thus, if the water table is above the ET extinction depth, the thickness of the lower layer is zero. If the water table is at the ground surface then the thickness of the upper layer is also zero. ET is only allowed from the upper of the two ET layers, if the lower layer exists.

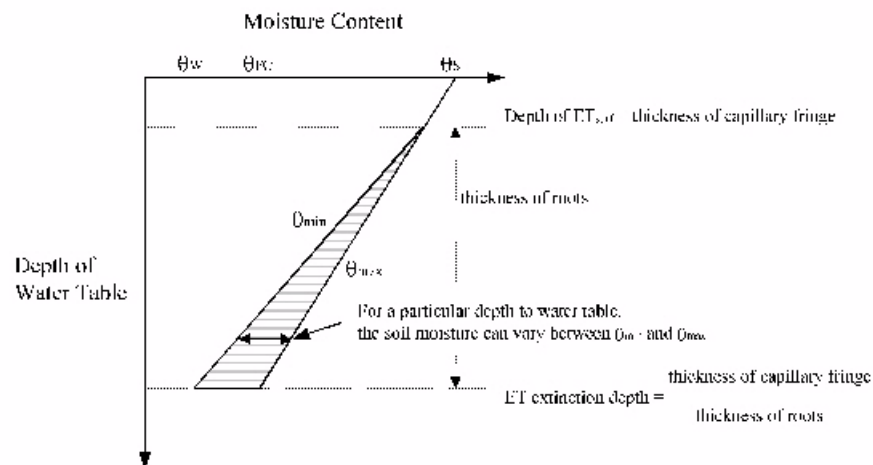


Figure 16.4 Allowable range for soil moisture in the upper ET layer for a given depth to water table.

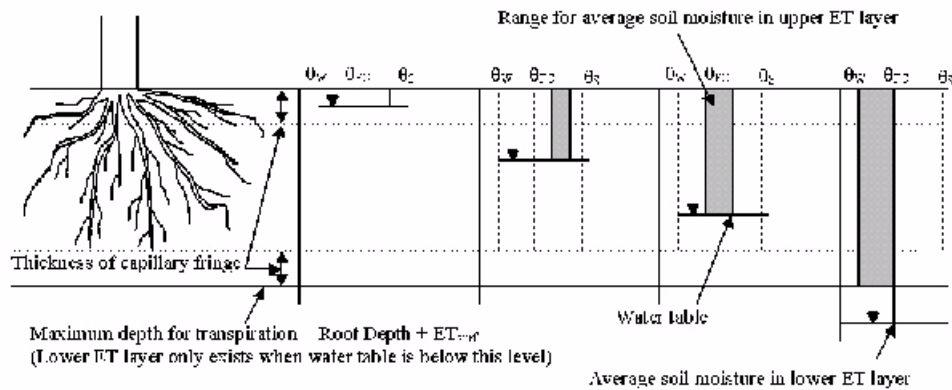


Figure 16.5 Soil moisture ranges for different depths to water table.

If the water table is at the ground surface then the moisture content equals the saturated moisture content, θ_{sat} , and all ET is taken from the saturated zone.

If the water table is below the ground surface, but above the ET surface, then the average moisture content of the ET layer is a linear function of the depth of the water table. That is, the average moisture content in the ET layer is lower when the water table is lower. If the water table is above the ET surface, the capillary fringe reaches the ground surface. In this case, the water content is not dependent on ET and any water lost to ET will be replaced from the groundwater table through capillary action.

If the water table is below the ET surface, but above the ET extinction depth, then the average water content will vary between a minimum, θ_{min} , and a maximum, θ_{max} . θ_{max} is the water content that would be present if no ET occurred. θ_{min} is the minimum water content that can exist in the upper ET layer when ET is active. Both θ_{min} and θ_{max} vary linearly with the depth to the water table. That is, θ_{min} and θ_{max} are lower when the water table is lower.

The difference between θ_{max} and the actual moisture content is the storage capacity of the unsaturated zone. Vertical infiltration to the saturated zone will only occur when the water content is equal to θ_{max} .

If the water table is below the ET extinction depth, then a lower ET layer exists. The moisture content of the lower ET layer is equal to the field capacity, which is the minimum water content when ET does not exist.



The average moisture content of the upper ET layer can range between the field capacity, θ_{FC} , and the wilting point, θ_{WP} , which is the minimum water content at which the plants can remove water from the soil.

16.3.2 Infiltration

At the beginning of each computational time step, rainfall first fills the interception storage. If I_{max} is exceeded, the excess water is added to the amount of ponded water on the ground surface, d_{oc} , which is the height of surface ponding before infiltration is subtracted.

Next, the maximum infiltration volume is limited by the rate of infiltration. Thus,

$$Inf_k = K_{inf} \cdot \Delta t$$

where Inf_k is the maximum amount of infiltration allowed during the time step due to the infiltration rate, K_{inf} is the infiltration rate and Δt is the calculation time-step.

The maximum infiltration volume is also limited by the available storage volume in the unsaturated zone, which is calculated by

$$Inf_v = (\theta_{sat} - \theta_{t-1}) \cdot z_{wt}$$

where θ_{sat} is the saturated water content, θ_{t-1} is the water content at the end of the previous time-step and z_{wt} is the depth of the water table.

The actual infiltration to the unsaturated zone, Inf_{actual} , is then calculated as the minimum of the amount of ponded water before infiltration, the rate limited amount of infiltration or the maximum volume of infiltration. Thus,

$$Inf_{actual} = \min(d_{oc}, Inf_k, Inf_v)$$

Subsequently d_{oc} and q_{act} are updated

$$d_{oc} = d_{oc}^* - I_{act} \quad [mm]$$

$$\theta_{act} = \theta_{act}^* - (I_{act} / (z_d \cdot 1000)) \quad [-]$$

where * refers to the parameter value before updating



16.4 Simplified Macropore Flow (bypass flow)

Flow through macropores in unsaturated soil is important for many soil types. In the Unsaturated Zone module, a simple empirical function is used to describe this process. The infiltration water is divided into one part that flows through the soil matrix and another part, which is routed directly to the groundwater table (bypass flow).

The bypass flow is calculated as a fraction of the net rainfall for each UZ time step. The actual bypass fraction is a function of a user-specified maximum fraction and the actual water content of the unsaturated zone, assuming that macropore flow occurs primarily in wet conditions.

Thus, the bypass flow, Q_{bypass} , is calculated as

$$Q_{bypass} = P_{net} P_{frac} \sqrt{\alpha_{10} \beta_{50}} / \Delta t \quad (16.28)$$

where P_{net} is the net rainfall rate, and P_{frac} is the maximum fraction of the net rainfall which can bypass the matrix (under wet conditions). α_{10} and β_{50} are used to reduce the total bypass fraction under dry conditions.

α_{10} and β_{50} are calculated internally by MIKE SHE and depend on the actual water contents of the unsaturated zone 10cm and 50cm below the ground surface, respectively. The relationship used to calculate α_{10} and β_{50} is illustrated in Figure 16.6. α_{10} and β_{50} vary linearly between 0.0 and 1.0 when the water content is between θ_2 and θ_1 . If the water content is below θ_2 , α_{10} and β_{50} equal 0.0. If the water content is above θ_1 , α_{10} and β_{50} equal 1.0.

Typically, macropore flow is highest in wet conditions when water is flowing freely in the soil (e.g. moisture content above the field capacity, θ_{FC}) and zero when the soil is very dry (e.g. moisture content at the wilting point, θ_{WP})

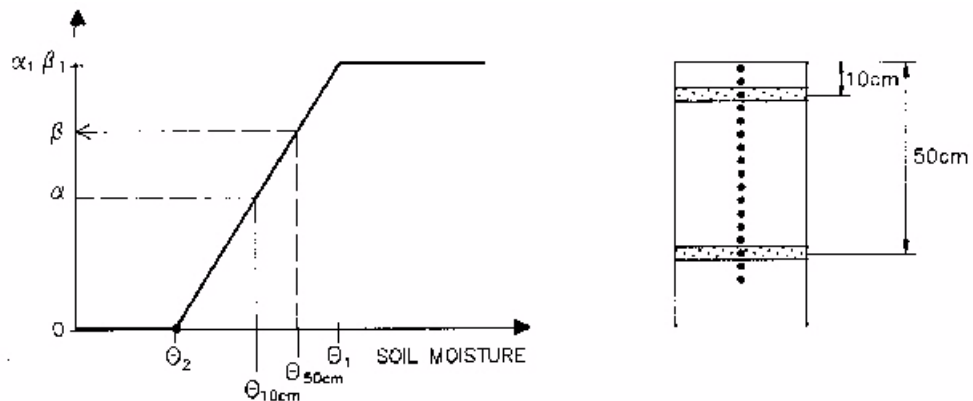


Figure 16.6 α and β as a function of the soil moisture content 10 cm and 50 cm below the ground surface, respectively.

16.5 Lumped UZ Calculations

In principle, unsaturated flow should be calculated individually for every ground surface node in the model domain. However, for large models the unsaturated flow calculations can become, by far, the most time consuming part of the solution. In particular, solving Richards' equation in every grid square can be quite time consuming when simulating long time series of rainfall.

To reduce the computational burden, it is possible to lump the unsaturated zone calculations where the unsaturated flow conditions are identical. The unsaturated flow conditions in two cells are identical when they have

- identical soil- and vegetation characteristics, AND
- identical boundary conditions.

If these two conditions are met, then the calculations need only be made in one of the cells and the results transferred to the other cell.

In practical terms, the first condition is usually not a serious restriction since most models are divided into several homogeneous soil zones. The second condition, however, is much more restrictive. Fluctuations in the groundwater table usually vary from cell to cell, and spatial variations in rainfall and the topography cause overland flow and infiltration to vary continuously across the domain.



However, if homogeneous zones can be defined based on

- Topography,
- Meteorology,
- Vegetation,
- Soil, and
- Bypass characteristics,

then a representative cell for the zone can be defined and used for the UZ-calculations. If this is done, then the boundary conditions from the representative cell (i.e. infiltration rate, evapotranspiration loss and groundwater recharge) can be transferred to the other cells within the zone.

Such an approximation does not introduce any water balance errors, but it can influence the dynamics of the simulation. However, an intelligent grouping of the cells can reduce computational burden considerably.

The initial definition of homogeneous zones can be made using the depth to the groundwater table and the soil, vegetation, and rainfall distributions. It is often necessary to re-group the columns several times during the calibration phase, until the groundwater regime is reasonably calibrated. Also, when the groundwater table is shallow, smaller intervals are usually required.

16.6 Coupling the Unsaturated Zone to the Saturated Zone

Briefly, the interaction between the unsaturated and saturated zones is solved by an iterative mass balance procedure, where the lower part of the unsaturated node system may be solved separately in a pseudo time step, between two real time steps. This coupling procedure ensures a realistic description of the water table fluctuations in situations with shallow soils. Particularly in these cases it is important to account for a variable specific yield above the water table, as the specific yield depends on the actual soil moisture profile and availability of that water.

The recharge to the groundwater is determined by the actual moisture distribution in the unsaturated zone. A correct description of the recharge process is rather complicated because the water table rises as water enters the saturated zone and affects flow conditions in the unsaturated zone. The actual rise of the groundwater table depends on the moisture profile above the water table, which is a function of the available unsaturated storage



and soil properties, and the amount of net groundwater flow (horizontal and vertical flow and source/sink terms).

The main difficulty in describing the linkage between the two the saturated (SZ) and unsaturated (UZ) zones arises from the fact that the two components (UZ and SZ) are explicitly coupled (i.e., run in parallel) and not solved in a single matrix with an implicit flux coupling of the UZ and SZ differential equations. Explicit coupling of the UZ and SZ modules is used in MIKE SHE to optimize the time steps used and allows use of time steps that are representative of the UZ (minutes to hours) and the SZ (hours to days) regimes. MIKE SHE overcomes problems associated with the explicit coupling of the UZ and SZ modules by employing an iterative procedure that conserves mass for the entire column by considering out-flows and source/sink terms in the saturated zone.

Error in the mass balance originates from two sources; 1) keeping the water table constant during a UZ time step and 2) using an incorrect estimate of the specific yield, S_y (the difference between the moisture content at saturation, θ_s , and moisture content at field capacity, θ_{fc}) in the SZ-calculations. This is illustrated in Figure 16.7a. If outflow from the SZ is neglected, it appears from the figure that during the time n to $n+m$, the column has lost V_1 mm of water (the light grey shaded area) and gained V_2 mm (the dark shaded area).

The changes calculated by the UZ module for the areas V_1 and V_2 represent a redistribution of water in the unsaturated zone to obtain an equilibrium moisture profile within the soil column. Comparing the equilibrium moisture content and the moisture content at UZ time n in Figure 16.7a shows that the moisture content is too high in the upper portions of the soil column. This should result in downward flow in the unsaturated zone, loss of soil moisture in area V_1 , increased soil moisture in area V_2 , and a rise in the water table. However, the SZ module uses a constant specific yield (S_y) defined for each grid cell in each calculation layer. On the other hand, the UZ can have a unique S_y value for each UZ node, which may differ from the S_y value used by the SZ. Thus, mass balance errors can occur in exchange calculations between the two modules. A mass-conservative solution is achieved by using a step-wise adjustment of the water table and recalculation of the UZ solution until the area of V_1 and V_2 are equal (see Figure 16.7b).

The procedure to deal with this mass balance error consists of a bookkeeping of the accumulated mass balance error, E_{cum} , for each UZ column and the upper SZ calculation cell associated with the column on a cell-by-cell basis. If $|E_{cum}|$ exceeds a user specified value, E_{max} , the UZ coupling



correction procedure corrects the water table of the upper SZ cell (i.e., the lower boundary condition for the UZ).

The correction procedure of step-wise water table adjustments and additional UZ calculations is repeated until $|E_{cum}|$ is less than E_{max} for each column that has failed the E_{max} criteria. During this correction procedure, the UZ module of MIKE SHE operates on a copy of the water table solution for the upper SZ calculation layer. After each SZ time step the UZ copy is updated with the new water table from the SZ solution and then adjusted during the succeeding UZ time step(s) until the next SZ time step. The calculated adjustments are converted to an additional flux term (multiplied with the specific yield of SZ and divided by the SZ time step length) and added to the uncorrected UZ-to-SZ flux term. The corrected UZ-to-SZ flux term is used by the SZ module as an explicit source/sink term during the next SZ time step.

The size of E_{max} determines the largest allowable mass balance before adjustments are made. Typically, an E_{max} value between 1-2 mm is an appropriate choice for regional MIKE SHE simulations. The E_{max} value is specified in the UZ Computational Control Parameters (V.2 p. 39) dialogue.

16.6.1 Steps in the Coupling Procedure

The following outlines the actual steps in the coupling procedure used for each UZ time step:

- 1 If the total water content above the datum Z_0 (Z_0 should always be lower than the lowest elevation of the water table) is designated W^n , the the UZ contribution to the E_{cum} error term in UZ-time step n to $n+1$ is

$$q_u^{n+1} = (W^{n+1} - W^n) / \Delta t^{n+1} + q_I + q_E \quad (16.29)$$

where W^{n+1} = the new water content; q_I = the infiltration rate (negative downwards); q_E = the evapotranspiration loss. (Note: negative values of q_u indicate downward flow).

- 2 Assuming that the groundwater outflow in a cell is steady, the accumulated error at UZ time $n+1$ is:

$$E_{cum}^{n+1} = \left(q_u^{n+1} + \frac{q_G^n \Delta_{n_G}}{\Delta t_{n_G+1}} + q_s^{n+1} \right) \Delta t + E_{cum}^n \quad (16.30)$$

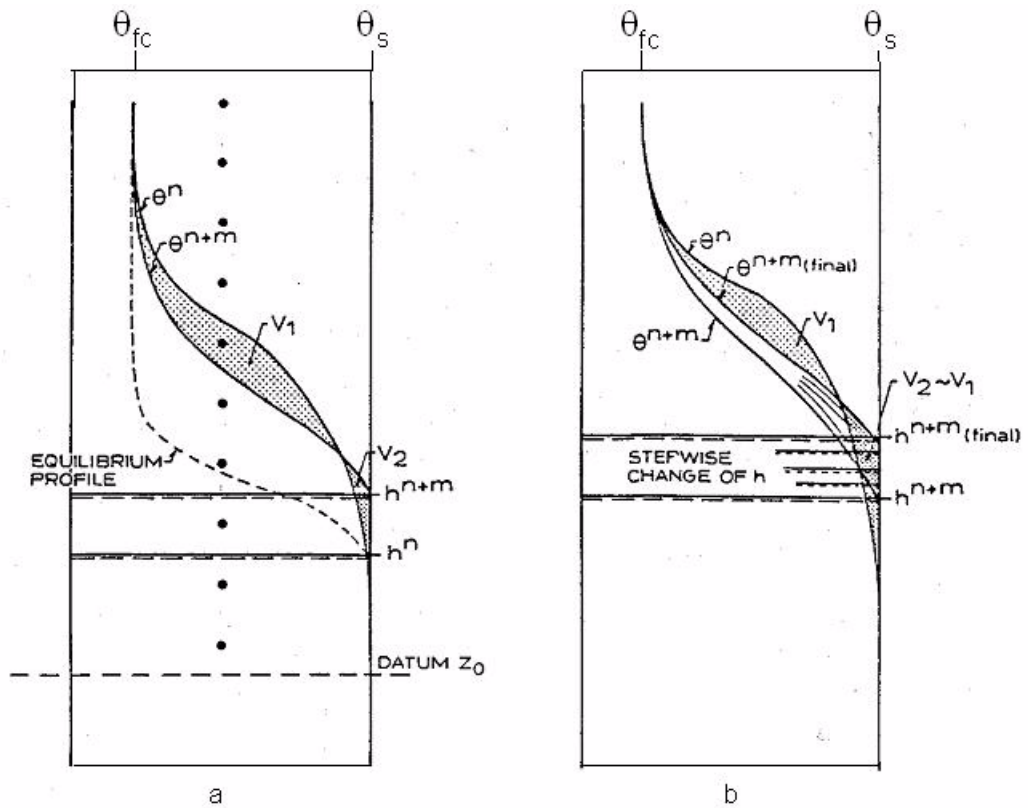


Figure 16.7 a) Soil moisture content at two times n and $n+m$ without corrections, and b) Soil moisture content at time $n+m$ before and after correction.

where $q_G^{n_G}$ (positive outwards) is the sum of the groundwater outflow rate for the cell in the last groundwater time step (n_G) scaled to the new SZ time step length (n_G+1) and q_s^{n+1} (positive outwards) is the sum of source sink terms calculated by the UZ module for the current time $n+1$ (e.g., stream/aquifer exchanges, irrigation).

It should be noted that if E_{cum} is less than zero there is a deficit of water stored in the column and if E_{cum} is greater than zero there is an excess of water stored in the column.

- 3 If $|E_{cum}^{n+1}|$ less than E_{max} corrections are not made for the current UZ time step.
- 4 If $|E_{cum}^{n+1}|$ exceeds E_{max} the following corrections are made:
 - a) If $|E_{cum}^{n+1}|$ is negative or positive, the water table is raised or lowered, respectively, in prescribed increments that depend on the dis-



tance between UZ nodes and the UZ-calculation in time step n to $n+1$ is repeated as described above.

b) In the Full Richards solution the UZ flow solution is repeated for the last three nodes above the water table to reduce numerical overhead. In the Gravity Flow option, the UZ flow solution is repeated for the entire column. The UZ flow solution is not repeated for the two-layer UZ option.

c) The change in water volume W^{n+1*} over the entire column is computed and a new $|E_{cum}^{n+1}|$ is calculated.

d) If $|E_{cum}^{n+1}|$ is less than aE_{max} , where a is a hard-coded correction factor equal to 0.9, the error associated with the solution is considered acceptable and the procedure stops. If $|E_{cum}^{n+1}|$ is greater than or equal to aE_{max} , the solution is unacceptable and steps a) through d) are repeated until criteria d) is satisfied. The value a defines a threshold for stopping the procedure lower than that used to initiate the procedure, which prevents correction overshoots.

e) If E_{cum}^{n+1} changes sign the solution is considered acceptable and the procedure stops. The adjustment required to obtain a $|E_{cum}^{n+1}|$ value of zero is calculated using a secant line approach.

5 A new recharge rate, q_u^{n+1*} is calculated taking the adjustments into account.

$$q_u^{n+1*} = q_u^{n+1} - (h^{n*} - h^n)S_y/\Delta t \quad (16.31)$$

where h^{n*} is the new water table elevation after step d) calculated by the UZ module and Δt is the length of the current UZ time step ($n+1$). If SZ outflows for the next SZ time step (n_G+1) are unchanged, the water table from the SZ calculation will be $h^{n+1} = h^{n*}$ calculated in the last UZ time step before an SZ time step (see Eq. (16.31)).

If $(h^{n*} - h^n)S_y/\Delta t > q_{max}$, where q_{max} is a maximum infiltration rate, the corrected rate is reduced to q_{max} . In the Richards Equation and Gravity Flow options q_{max} is $0.7K_s$ and $0.4K_s$ for rising and falling water table conditions, respectively, where K_s is the saturated hydraulic conductivity of the UZ node at the water table. In the Two-layer UZ option, the infiltration rate is used to constrain the corrected rate.



Steps 1-5 are repeated for all UZ time steps within each SZ time step. The flows are accumulated and passed as an average rate, q_u , for the next SZ time step. The average q_u is used as a flux boundary condition in the SZ differential equations.

16.6.2 Limitations

The coupling between UZ and SZ is limited to the upper most calculation layer of the saturated zone. The unsaturated zone profiles should extend below the maximum depth of the water table in top SZ calculation layer, or to the bottom of uppermost SZ calculation layer. If the top layer of the SZ model dries out, then the UZ model usually assumes a lower pressure head boundary equal to the bottom of the uppermost SZ layer. For more detailed information on the UZ-SZ bottom boundary see Lower Boundary (p. 269)

16.6.3 Evaluation of the UZ-SZ Coupling

The WM_Print log file generated by MIKE SHE should be reviewed after each simulation to evaluate the performance of the UZ module. If the user specified maximum UZ iterations is exceeded an excessive number of times and there are no problems with the soil data used in the UZ module, the UZ and SZ time step should be evaluated. Sometimes it is possible to reduce the number of times the maximum UZ iterations is exceeded by making the UZ and SZ time steps more similar. Typically the SZ to UZ time step ratio should be no larger than four.

It is also useful to save the value of E_{cum} as a grid series output, or as a detailed time series output at critical locations. These plots can be used to determine if there are locations or periods of time during the simulation where the E_{cum} term exceeds E_{max} . This can occur if

- the water table drops below the first SZ calculation layer (positive value),
- the water table rises above the top of the first SZ calculation layer (negative value),
- the vertical hydraulic conductivity in the upper SZ calculation layer is much greater than the saturated hydraulic conductivity used in the UZ, or if
- the drainage time constant is too high.

In the first two cases above, the *epsilon* term can exceed E_{max} because the UZ module cannot get rid of *epsilon* because there is no available storage for the error term. In the third case, the UZ and SZ hydraulic properties should be consistent or it will be difficult for MIKE SHE to simulate con-



sistent vertical flow rates. In the last case, the drainage time constant should be reduced to prevent excessive and unrealistic drainage outflows from the SZ module.



17 SATURATED FLOW - REFERENCE

The Saturated Zone (SZ) component of MIKE SHE WM calculates the saturated subsurface flow in the catchment. MIKE SHE allows for a fully three-dimensional flow in a heterogeneous aquifer with shifting conditions between unconfined and confined conditions.

The spatial and temporal variations of the dependent variable (the hydraulic head) is described mathematically by the 3-dimensional Darcy equation and solved numerically by an iterative implicit finite difference technique.

MIKE SHE gives the opportunity to choose between two groundwater solvers - the SOR groundwater solver based on a successive over-relaxation solution technique and the PCG groundwater solver based on a preconditioned conjugate gradient solution technique. The formulation of potential flow and sink/source terms differs between the two modules to some extent.

The Saturated Zone Component interacts with the other components of MIKE SHE WM mainly by using the boundary flows from other components implicitly or explicitly as sources and sinks.

17.1 3D Finite Difference Method

The governing flow equation for three-dimensional saturated flow in saturated porous media is:

$$\frac{\partial}{\partial x} \left(K_{xx} \frac{\partial h}{\partial x} \right) + \frac{\partial}{\partial y} \left(K_{yy} \frac{\partial h}{\partial y} \right) + \frac{\partial}{\partial z} \left(K_{zz} \frac{\partial h}{\partial z} \right) - Q = S \frac{\partial h}{\partial t} \quad (17.1)$$

where K_{xx}, K_{yy}, K_{zz} the hydraulic conductivity along the x, y and z axes of the model, which are assumed to be parallel to the principle axes of hydraulic conductivity tensor, h is the hydraulic head, Q represents the source/sink terms, and S_s is the specific storage coefficient.

Two special features of this apparently straightforward elliptic equation should be noted. First, the equations are non-linear when flow is unconfined and, second, the storage coefficient is not constant but switches between the specific storage coefficient for confined conditions and the specific yield for unconfined conditions.



17.1.1 The Pre-conditioned Conjugate Gradient (PCG) Solver

As an alternative to the SOR-solver, MIKE SHE's groundwater component also includes the pre-conditioned conjugate gradient solver (Hill, 1990). The PCG solver includes both an inner iteration loop, where the head dependent boundaries are kept constant, and an outer iteration loop where the (non-linear) head dependent terms are updated. The PCG solver includes a number of additional solver options that are used to improve convergence of the solver. The default values will generally ensure good performance. For the majority of applications, there is no need to adjust the default solver settings. If, on the other hand, non-convergence or extremely slow convergence is encountered in the SZ component, then some adjustment of the solver settings may help.

The PCG solver in MIKE SHE, which is identical to the one used in MODFLOW (McDonald and Harbaugh, 1988), requires a slightly different formulation of the hydraulic terms when compared to the SOR solver.

Potential flow terms

The potential flow is calculated using Darcy's law

$$Q = \Delta h C \quad (17.2)$$

where Δh is the piezometric head difference and C is the conductance.

The horizontal conductance in Eq. (17.2) is derived from the harmonic mean of the horizontal conductivity and the geometric mean of the layer thickness. Thus, the horizontal conductance between node i and node $i-1$ will be

$$C_{i-1/2} = \frac{KH_{i-1,j,k} KH_{i,j,k} (\Delta z_{i-1,j,k} + \Delta z_{i,j,k})}{(KH_{i-1,j,k} + KH_{i,j,k})} \quad (17.3)$$

where, KH is the horizontal hydraulic conductivity of the cell and Δz is the saturated layer thickness of the cell.

The vertical conductance between two cells is computed as a weighted serial connection of the hydraulic conductivity, calculated from the middle of layer k to the middle of the layer $k+1$. Thus,

$$C_v = \frac{\Delta x^2}{\frac{\Delta z_k}{2K_{z,k}} + \frac{\Delta z_{k+1}}{2K_{z,k+1}}} \quad (17.4)$$



where Δz is the layer thickness

Dewatering conditions

Consider the situation in Figure 17.1, where the cell below becomes dewatered.

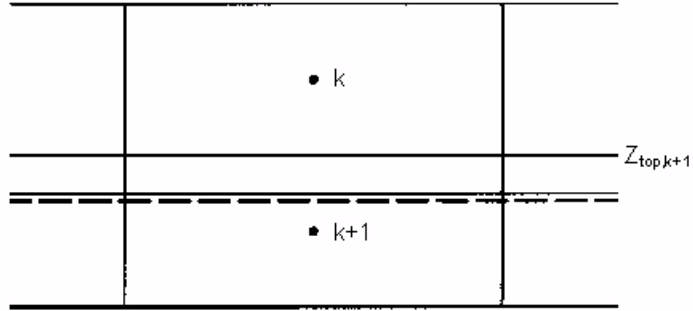


Figure 17.1 Dewatering conditions in a lower cell.

The actual flow between cell k and $k+1$ is

$$q_{k+1/2} = Cv_{k+1/2}(z_{top,k+1} - h_k) \quad (17.5)$$

In the present solution scheme the flow will be computed as

$$q_{k+1/2} = Cv_{k+1/2}(h_{k+1} - h_k) \quad (17.6)$$

Subtracting Eqs.(17.5) from (17.6) gives the correction term

$$q_c = Cv_{k+1/2}(h_{k+1} - z_{top,k+1}) \quad (17.7)$$

which is added to the right-hand side of the finite difference equation using the last computed head.

A correction must also be applied to the finite difference equation if the cell above becomes dewatered.

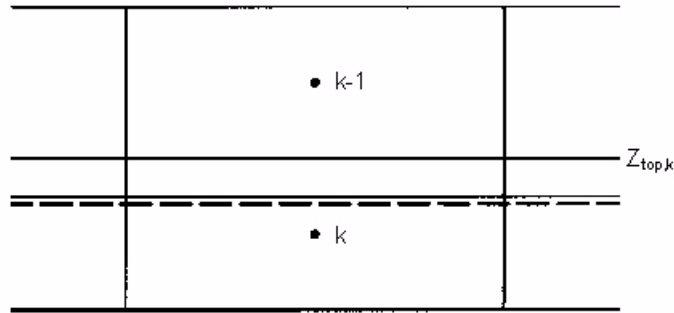


Figure 17.2 Dewatering conditions in the cell above

Thus, from Figure 17.2, the flow from cell $k-1$ to k is

$$q_{k-1/2} = Cv_{k-1/2}(h_{k-1} - z_{top,k}) \quad (17.8)$$

where, again the computed flow is

$$q_{k-1/2} = Cv_{k-1/2}(h_{k-1} - h_k) \quad (17.9)$$

Subtracting Eqs. (17.8) from (17.9) gives the correction term

$$q_c = Cv_{k-1/2}(z_{top,k} - h_k) \quad (17.10)$$

which is added to the right-hand side of the finite difference equation using the last computed head

Storage terms

The storage capacity is computed by

$$\frac{\Delta w}{\Delta t} = \frac{S2(h^n - z_{top}) + S1(z_{top} - h^{n-1})}{\Delta t} \quad (17.11)$$

where n is time step, $S1$ is the storage capacity at the start of the iteration at time step n , and $S2$ is the storage capacity at the last iteration.

For confined cells the storage capacity is given as

$$S = \Delta x^2 \Delta z S_{art} \quad (17.12)$$



and for unconfined cells the storage capacity is given as

$$S = \Delta x^2 S_{free} \quad (17.13)$$

Maximum Residual Error

The maximum residual error is the largest allowable value of residual error during an iteration. The solution is obtained when the residual error during an iteration in any computational node is less than the specified tolerance.

The value of the maximum residual error should be selected according to aquifer properties and the dimensions of the model. In practice the maximum residual error value will always be a compromise between accuracy and computing time. It is recommended to check the water balance carefully at the end of the simulation, but it should be emphasized that large internal water balance errors between adjacent computational nodes may not be detected. If large errors in the water balance are produced the maximum residual error should be reduced.

Gradual activation of SZ drainage

To prevent numerical oscillations the drainage constant may be adjusted between 0 and the actual drainage time constant defined in the input for SZ drainage. The option has been found to have a dampening effect when the groundwater table fluctuates around the drainage level between iterations (and does not entail reductions in the drain flow in the final solution).

Mean values of horizontal SZ-conductance:

To prevent potential oscillations of the numerical scheme when rapid changes between dry and wet conditions occur a mean conductance is applied by taking the conductance of the previous (outer) iteration into account.

Under relaxation

The PCG solver can optionally use an under-relaxation factor between 0.0 and 1.0 to improve convergence. In general a low value will lead to convergence, but at a slower convergence rate (i.e. with many SZ iterations). Higher values will increase the convergence rate, but at the risk of non-convergence.

Automatic (dynamic) estimation of under-relaxation factors

If the automatic estimation of the under-relaxation factor is allowed, the under-relaxation factor is calculated automatically as part of the outer iteration.



ation loop in the PCG solver. The algorithm determines the factors based on the minimum residual-2-norm value found for 4 different factors. To avoid numerical oscillations the factor is set to 90% of the factor used in the previous iteration and 10% of the current optimal factor.

The time used for automatic estimation of relaxation factors may be significant when compared to subsequently solving the equations and the option is only recommended in steady-state cases.

Under-relaxation by user-defined constant factor

This option allows the user to define a constant relaxation factor between 0.0 and 1.0. In general a value of 0.2 has been found suitable for most simulations.

2-norm reduction-criteria in the inner iteration loop

When the 2-norm option is active, the inner iteration loop of the PCG solver ends when the specified reduction of the 2-norm value is reached. Thus, if the 2-norm reduction criteria is set to 0.01, the inner iteration residual must be reduced by 99% before the inner iteration loop will exit. This option is sometimes efficient in achieving convergence in the linear matrix solution before updating the non-linear terms in the outer iteration loop. It may thus improve the convergence rate of the solver. Continued iterations to meet user-defined criteria in the inner loop may not be feasible before the changes in the outer iteration loop have been minimised. On the other hand, very few iterations in the inner loop may not be sufficient. The 2-norm may be used to achieve a more optimal balance between the computational efforts spent in the respective solver loops.

Convergence is, however, not assumed until the user defined head and water balance criteria are fulfilled. A reasonable value for the 2-norm reduction criteria has been found to be 0.01.

17.1.2 PCG Steady State Solver

The PCG Steady-State Solver is virtually identical to the transient PCG solver but has been implemented separately to enhance efficiency. In particular, experience has shown that different solver settings may be required when solving the system in steady state versus a transient solution. Furthermore, since the solvers have been implemented separately, there are a couple of options in the steady-state solver that are not available in the transient solver.

Canyon exchange option

The Canyon exchange option is only available in the steady state PCG solver. It can be used to describe the exchange between a groundwater



aquifer and a river when the river cuts deeply into the aquifer (e.g. through a narrow valley). If the river water level is below the bottom of the adjacent computational groundwater layer, the potential head gradient is reduced. In this case, the head difference in layers above the river level is limited by the bottom elevation of the layer. Thus,

$$\Delta h_i = h_i - \max(h_{riv}, z_i) \quad (17.14)$$

where h_i is the head in the adjacent groundwater node, h_{riv} is the head in the river, and z_i is the bottom of the current layer.

Without the 'Canyon' option, MIKE SHE effectively assumes that the river is hydraulically connected to the upper most model layer, since MIKE SHE calculates the exchange flow with all layers that intersect the river based on the difference between the river level and the water table.

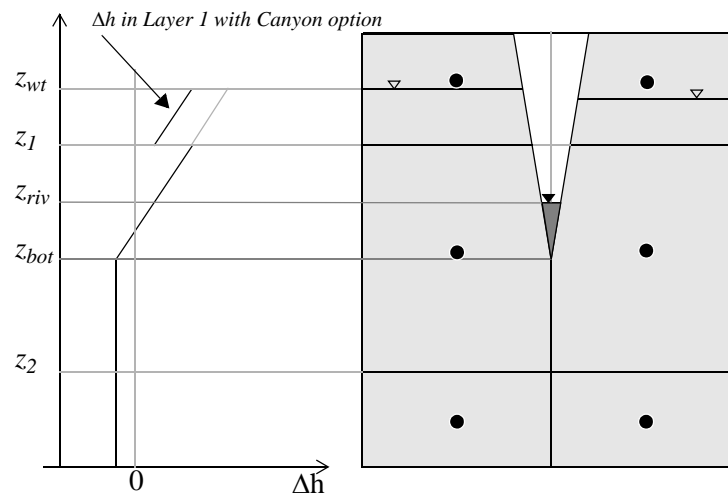


Figure 17.3 Water table elevation versus the head difference using Eqs. (14.2), (14.3), and (14.7).

Average steady-state river conductance option

In the steady-state PCG solver, the conductance used for the SZ-river exchange in each iteration is averaged with the conductance of the previous iteration. This is done to reduce the risk of numerical instabilities when the conditions are changing between flow/no flow conditions in a computational cell. It is recommended to use this option, as it tends to enhance convergence.



Steady-state (constant) river water depth

When running a steady-state simulation that includes SZ-river exchange, a constant water depth may be specified for the river network, which can be used to calculate the head gradient driving the exchange flow.

If a constant river water depth is not specified, then the river water levels are determined in the following order

- 1 MIKE SHE hot-start water level, if MIKE SHE hot-start is specified
- 2 Initial water levels from MIKE 11, if the MIKE 11 coupling is used
- 3 Water level equal to the river bed if not 1) or 2) - (i.e. dry river - no flow from the river to the aquifer)

17.1.3 Boundary Conditions

The SZ module supports the following three types of boundary conditions:

- 1 Dirichlet conditions, (Type 1) where the hydraulic head is prescribed on the boundary
- 2 Neumann conditions, (Type 2) where the gradient of the hydraulic head (flux) across the boundary is prescribed
- 3 Fourier conditions, (Type 3) where the head dependent flux is prescribed on the boundary.

The head can be prescribed for all grid nodes (i.e. at the catchment boundary, as well as inside the model area) and for all computational layers. The head may be time-invariant equal to the initial head or can vary in time as specified by the user. An important option is the transfer of space- and time interpolated head boundaries from a larger model to a sub-area model with a finer discretisation.

Prescribed gradients and fluxes can be specified in all layers at the model boundary. Sinks and sources in terms of pumping or injection rates can be specified in all internal nodes. If the unsaturated zone component is not included in the model, the ground water recharge can be specified.

The exchange flow to the river system is included in the source/sink terms and can be regarded as a Type 3 boundary condition for cells with 'contact' to the river system. The exchange flow is a function of the water level in the river, the river width, the elevation of the riverbed, as well as the hydraulic properties at the riverbed and aquifer material.



Distribution of fluxes to the internal cells

Flux boundary

The total discharge of the actual time step (constant or time-varying) is distributed over the internal cells as a function of horizontal conductivities, cell sizes (constant in MIKE SHE) and full or saturated thicknesses (inflow / outflow, respectively).

Inflow (positive discharge):

Cell Factor = Conductivity X Size X Full layer thickness / $\sum_{i=1,n} (\text{Size}_i \times \text{Coni} \times \text{Thick}_i)$

Cell Discharge $Q_i = Q_{\text{tot}} \times \text{Cell Factor}_i$

Outflow (negative discharge):

Same as above, but using actual saturated thickness instead of full layer thickness. In the case that all cells are dry the solution will switch to full layer thickness. However, in this case, the solution will be unstable anyway.

Gradient boundary

Each cell receives a discharge calculated from the actual gradient (constant or time-varying), conductivity, cell size and saturated thickness. Positive gradient yields inflow:

$Q = \text{Gradient} \times \text{Conductivity} \times \text{Size} \times \text{Saturated thickness}$

Notes on Grid size

Distribution of actual cell discharges to the X- and Y- flow velocities are stored on the MIKE SHE result file, where they are needed for presentation and water balance extraction.

The X/Y-flow stored in a cell represents the flow from this cell to the neighbour cell in positive X/Y direction. The discharge of a cell in contact with more than one boundary cells of the actual boundary section is distributed as X/Y flow components to/from these boundary cells proportional to the grid size of the flow surfaces (constant in MIKE SHE).

Notes on the PCG solver

The discharges to/from the internal cells are updated at the start of each outer iteration and added to the “Right-Hand-Side” of the PCG solver. The



discharges are distributed to the X/Y flow velocities (for results storing) at the end of each SZ time step.

Notes on the SOR solver

The discharges to/from the internal cells are updated at the start of each iteration and used as point source/sink terms in the solution. The discharges are distributed to the X/Y flow velocities (for results storing) at the end of each SZ time step.

Saturated Zone Drainage

The MIKE SHE allows for flow through drains in the soil. Drainage flow occurs in the layer of the ground water model where the drain level is located. In MIKE SHE the drainage system is conceptually modelled as one 'big' drain within a grid square. The outflow depends on the height of the water table above the drain level and a specified time constant, and is computed as a linear reservoir. The time constant characterises the density of the drainage system and the permeability conditions around the drains.

The drainage option may not only be used to simulate flow through drain-pipes, but also in a conceptual mode to simulate saturated zone drainage to ditches and other surface drainage features. The drainage flow simulates the relatively fast surface runoff when the spatial resolution of the individual grid squares is too large to represent small scale variations in the topography.

Drainage water can be routed to local depressions, rivers or model boundaries. See Groundwater Drainage (*V.1 p. 53*) for further details about routing of drainage water.

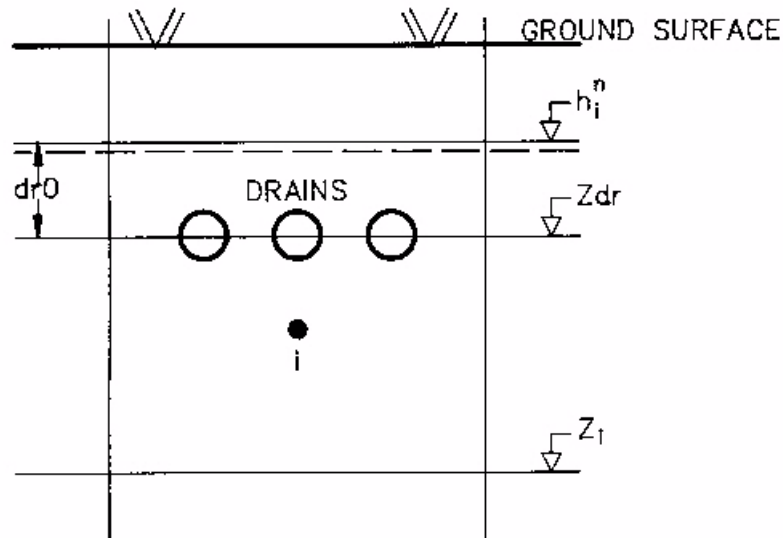


Figure 17.4 Schematic presentation of drains in the drainage flow computations.

Drainage with the PCG solver

When the PCG solver is used, the drain flow is added directly in the matrix calculations as a head dependent boundary and solved implicitly, by

$$q = (h - Z_{dr})C_{dr} \quad (17.15)$$

where h head in drain cell, Z_{dr} is the drainage level and C_{dr} is the drain conductance or time constant.

Exchange with surface water

The recharge/discharge to the surface water is depended on the components included in the simulation. The saturated zone component calculate the surface recharge/discharge in point with no unsaturated zone, that is if the unsaturated zone component is excluded or the piezometric head is above surface level.

The exchange between SZ and OL is carried out implicitly by constantly updating the overland water depth and is calculated by use of the Darcy equation.

$$Q = \Delta h C_{1/2}$$



where $C_{1/2}$ is the conductance from surface level to the middle of the top calculation layer.

In case of full contact between overland and the saturated zone the conductance between overland and layer 1 is computed as

$$C_{1/2} = \frac{\frac{\Delta x^2}{\Delta z}}{2K_z} \quad (17.16)$$

where Δz is the thickness of layer 1 and K_z is the vertical conductivity of layer 1.

In areas with reduced contact between overland and the saturated zone, the conductance between the overland flow and layer 1 is calculated by

$$C_{1/2} = \frac{\frac{\Delta x^2}{\Delta z}}{\frac{2K_z}{K_{leak}} + 1} \quad (17.17)$$

where Δz is the thickness of layer 1 and K_z is the vertical conductivity of layer 1 and K_{leak} is the specified leakage coefficient.

17.1.4 Initial Conditions

The initial conditions are specified in the Setup Editor and can be either constant for the domain or distributed, using .dfs2 or .shp files. The initial conditions in boundary cells are held constant during the simulation, which means that the initial head in cells with Dirichlet's boundary conditions is the boundary head for the simulation.

17.1.5 The Sheet Pile Module

The Sheet Piling module is an add-on module for the PCG groundwater (SZ) solver in MIKE SHE that allows you to reduce the conductance between cells in both the horizontal and vertical directions. Vertical sheet piling reduces the horizontal flow between adjacent cells in the x- or y-direction. It is defined by a surface with a specified leakage coefficient located between two MIKE SHE grid cells in a specified computational



layer. If the Sheet Piling module is active, then the horizontal conductance, C_h , for flow between two cells is calculated as

$$C_h = \frac{1}{\frac{1}{2 \cdot k_1 \cdot dz_1} + \frac{1}{k_{leak} \cdot dx \cdot dz^*} + \frac{1}{2 \cdot k_2 \cdot dz_2}} \quad (17.18)$$

where k_i is the hydraulic conductivity of the cells on either side of the sheet pile [L/T], k_{leak} is the leakage coefficient of the sheet pile between the cells [1/L], dz_i saturated layer thickness [L], and dz^* maximum of dz_1 and dz_2 [L]

Similarly, reducing the vertical conductance between two layers can simulate a restriction in vertical flow (e.g. thin clay layer or liner). Thus, if the Sheet Piling module is active for vertical flow, the vertical conductance between two layers is calculated as

$$C_v = \frac{dx \cdot dx}{\frac{dz_1}{2 \cdot k_1} + \frac{1}{k_{leak}} + \frac{dz_2}{2 \cdot k_2}} \quad (17.19)$$

where k_i is the hydraulic conductivity of the cells above and below the sheet pile [L/T], k_{leak} is the leakage coefficient [1/T], and dz_i is the layer thickness [L].

17.1.6 The Successive Overrelaxation (SOR) Solver

Numerical formulation

Equation (17.1) is solved by approximating it to a set of finite difference equations by applying Darcy's law in combination with the mass balance equation for each computational node.

Considering a node i inside the model area, the total inflow ΣQ_{ij}^{n+1} from neighbouring nodes and source/sinks between time n and time $n+1$ is given by:

$$\Sigma Q_{ij}^{n+1} = \Sigma q_z^{n+1} + \Sigma q_x^{n+1} + RH_i \Delta x^2 \quad (17.20)$$

where q_z^{n+1} is the volumetric flow in vertical direction, q_x^{n+1} is the volumetric flow in horizontal directions, R is the volumetric flow rate per unit volume from any sources and sinks, Δx is the spatial resolution in the hor-



horizontal direction and H_i is either the saturated depth for unconfined layers or the layer thickness for confined layers. See Figure 17.5 and Figure 17.6 for a description of the geometric relationships between the cells.

The horizontal flow components in Eq. (17.20) are given by

$$q_x^{n+1} = C\Delta h^{n+1} \quad (17.21)$$

where C is the horizontal conductance between any of the adjacent nodes in the horizontal directions.

The horizontal conductance in Eq. (17.21) is derived from the harmonic mean of the horizontal conductivity and the geometric mean of the layer thickness. Thus, the horizontal conductance between node i and node $i-1$ will be

$$C_{i-1/2} = \frac{KH_{i-1,j,k} KH_{i,j,k} (\Delta z_{i-1,j,k} + \Delta z_{i,j,k})}{(KH_{i-1,j,k} + KH_{i,j,k})} \quad (17.22)$$

where, KH is the horizontal hydraulic conductivity of the cell and Δz is the saturated layer thickness of the cell.

The vertical flow components in Eq. (17.20) are given by

$$q_z^{n+1} = \frac{K_v \Delta x^2 \Delta h^{n+1}}{\Delta z_{i,j,k} + \Delta z_{i,j,k+1}} \quad (17.23)$$

where K_v is the average vertical hydraulic conductivity between nodes in the vertical direction, and $\Delta z_{i,j,k}$ and $\Delta z_{i,j,k+1}$ are the saturated thicknesses of layers k and $k+1$ respectively. For the average vertical hydraulic conductivity, K_v , the SOR solver distinguishes between conditions where the hydraulic conductivity of layer k is greater than or less than the hydraulic conductivity of layer $k+1$. These cases are shown in Table 17.1 and Table 17.2.

Table 17.1 For the case when $K_k < K_{k+1}$.

		Layer k	
		confined	unconfined
Layer $k+1$	confined	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{\Delta z_{k+1}}{K_{k+1}}}$	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{\Delta z_{k+1}}{K_{k+1}} + \frac{z_{k+1} - h_k}{K_k}}$
	unconfined	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{h_{k+1} - z_{k+1}}{K_{k+1}}}$	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{h_{k+1} - z_{k+1}}{K_{k+1}} + \frac{z_{k+1} - h_k}{K_k}}$
		$\Delta h = h_k - h_{k+1}$	

Table 17.2 For the case when $K_k > K_{k+1}$.

		Layer k	
		confined	unconfined
Layer $k+1$	confined	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{\Delta z_{k+1}}{K_{k+1}}}$ $\Delta h = h_k - h_{k+1}$	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{\Delta z_{k+1}}{K_{k+1}}}$ $\Delta h = z_{k+1} - h_{k+1}$
	unconfined	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{h_{k+1} - z_{k+1}}{K_{k+1}}}$ $\Delta h = h_k - h_{k+1}$	$K_v = \frac{\Delta z_k + \Delta z_{k+1}}{\frac{h_{k+1} - z_{k+1}}{K_{k+1}}}$ $\Delta h = z_{k+1} - h_{k+1}$

The transient flow equation yields the finite difference expression

$$S \frac{h_i^{n+1} - h_i^n}{\Delta t} = \Sigma Q_{ij}^{n+1} \quad (17.24)$$

where S is the storage coefficient and Δt is the time step. Eq. (17.24) is written for all internal nodes N yielding a linear system of N equations with N unknowns. The matrix is solved iteratively using a modified Gauss Seidel method (Thomas, 1973).

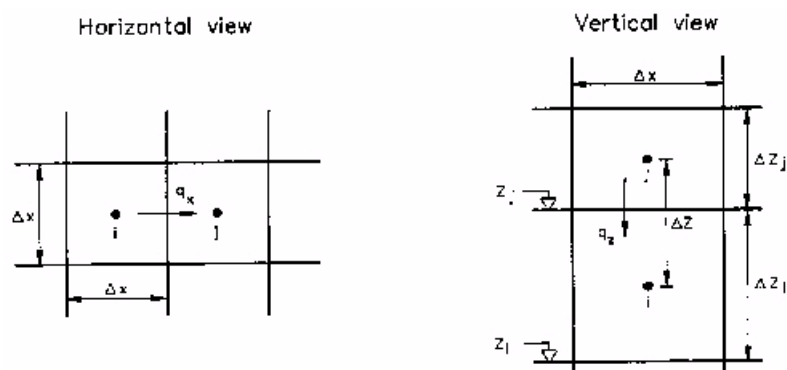


Figure 17.5 Spatial discretisation.

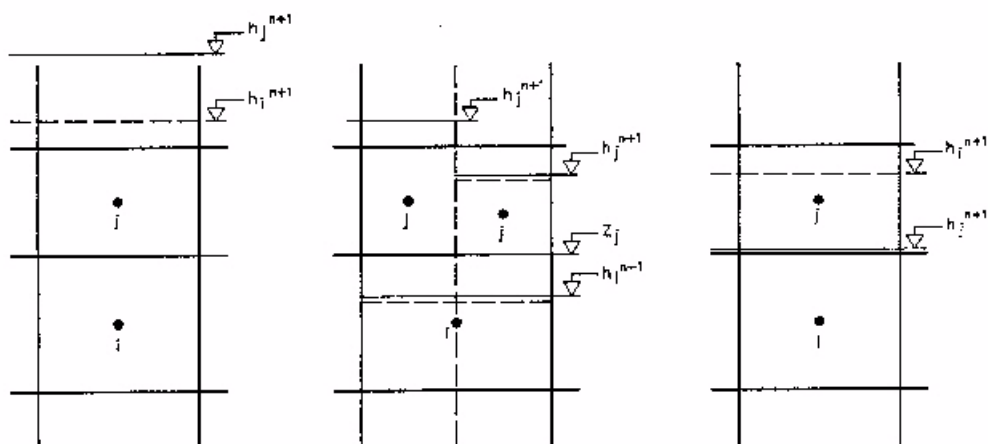


Figure 17.6 Types of vertical flow condition, a) confined conditions in nodes i and j , b) unconfined condition in node i , c) unconfined in nodes i and j , d) dry conditions in node j and confined conditions in node i .

Relaxation Coefficient

The relaxation coefficient, w , is used in the solution scheme to amplify the change in the dependent variable (hydraulic head h) during the iteration. The value of w should be less than 2 to ensure convergence but larger than 1 to accelerate the convergence.



The optimal value is the value for which the minimum number of iterations are required to obtain the desired tolerance. It is a complex function of the geometry of the grid and aquifer properties. Figure 17.7 illustrates the relation between w and the number of iterations for a given grid. In practice the optimal value of w can be found after setting up the grid. The model is run for a few time steps (e.g. ten) with a range of w values between 1 and 2, and the total number of iterations is plotted for each run against the w value as shown in Figure 17.7. The minimum number of iterations corresponds to the optimal value of w .

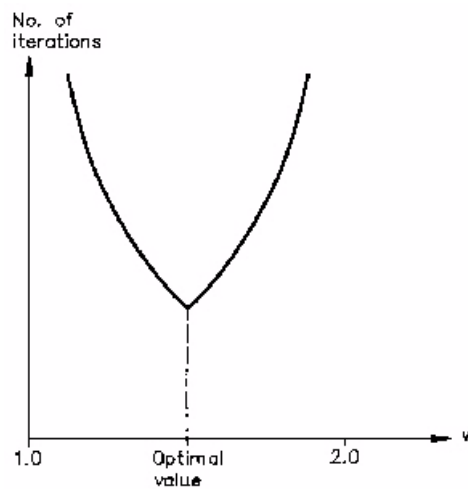


Figure 17.7 Empirically relationship between the relaxation coefficient w and number of iterations for a given model.

Maximum Residual Error

The maximum residual error is the largest allowable value of residual error during an iteration. The solution is obtained when the residual error during an iteration in any computational node is less than the specified tolerance.

The value of the maximum residual error should be selected according to aquifer properties and dimensions of the model. In practice, the maximum residual error value will always be a compromise between accuracy and computing time. It is recommended to check the water balance carefully at the end of the simulation, but it should be emphasized that large internal water balance errors between adjacent computational nodes may not be



detected. If large errors in the water balance are produced the maximum residual error should be reduced.

Drainage with the SOR Solver

When the SOR solver is used, drainage is only allowed from the top layer of the saturated zone model. In this case, the new water table position at the end of the time step is calculated from the flow balance equation

$$\Delta S = (Q_{dr} + \Sigma q)\Delta t \quad (17.25)$$

where ΔS is the storage change as a result of a drop in the water table, Q_{dr} is the outflow through the drain and Σq represents all other flow terms in a computational node in the top layer (i.e. net outflow to neighbouring nodes, recharge, evapotranspiration, pumping and exchange to the river etc.).

The change in storage per unit area can also be calculated from

$$\Delta S = (d_0 - d_t)S_y \quad (17.26)$$

where d_0 is the depth of water above the drain at the beginning of the time step, d_t is the depth of water above the drain at the end of the time step and S_y is the specific yield.

Q_{dr} is calculated based on the mean depth of water in the drain during the time step. Thus,

$$Q_{dr} = C_{dr} \cdot \frac{d_0 - d_t}{2} \quad (17.27)$$

where C_{dr} is the drain conductance or time constant.

Substituting (17.26) and (17.27) into (17.25) and rearranging, the water depth at the end of a time step, d_t , can be calculated by

$$d_t = \frac{\left[d_0 \left(S_y - \frac{C_{dr}\Delta t}{2} \right) - \Sigma q \Delta t \right]}{S_y + \frac{C_{dr}\Delta t}{2}} \quad (17.28)$$



From which the new water table elevation, h_t , at the end of the time step can be calculated by

$$h_t = Z_{dr} + d_t \quad (17.29)$$

where Z_{dr} is the elevation of the drain.

The drainage outflow is added as a sink term using the hydraulic head explicitly. The computation for drainage flow uses the UZ time step, which is usually smaller than the SZ time step. The initial drainage depth d_0 at the beginning of an SZ time step is set equal to $h - Z_{dr}$, where h is the water table elevation at the end of the previous SZ time step. d_0 is adjusted during the sequence of smaller time steps so that a successive lowering of the water table and the outflow occurs during an SZ time step. This approach often overcomes numerical problems when large time steps are selected by the user. If the drainage depth becomes zero during the calculations drainage flow stops until the water table rises again above the drain elevation.

17.2 Linear Reservoir Method

The linear reservoir module for the saturated zone in MIKE SHE was developed to provide an alternative to the physically based, fully distributed model approach. In many cases, the complexity of a natural catchment area poses a problem with respect to data availability, parameter estimation and computational requirements. In developing countries, in particular, very limited information on catchment characteristics is available. Satellite data may increasingly provide surface data estimates for vegetation cover, soil moisture, snow cover and evaporation in a catchment. However, subsurface information is generally very sparse. In many cases, subsurface flow can be described satisfactorily by a lumped conceptual approach such as the linear reservoir method.

The MIKE SHE modelling system used with the linear reservoir module for the saturated zone may be viewed as a compromise between limitations on data availability, the complexity of hydrological response at the catchment scale, and the advantages of model simplicity. The combined lumped/physically distributed model was primarily developed to provide a reliable, efficient instrument in the following fields of application:

- Assessment of water balance and simulation of runoff for ungauged catchments
- Prediction of hydrological effects of land use changes



- Flood prediction

The following sections first provide an overview of the linear reservoir theory, followed by detailed descriptions of the implementation in MIKE SHE.

17.2.1 Linear Reservoir Theory

A linear reservoir is one, whose storage is linearly related to the output by a storage constant with the dimension time, also called a time constant, as follows:

$$S = kQ \quad (17.30)$$

where S is storage in the reservoir, k is the time constant, and Q is the outflow rate from the reservoir.

The concept of a linear reservoir was first introduced by Zoch (1934,1936,1937) in an analysis of the rainfall and runoff relationship. Also, a single linear reservoir is a special case of the Muskingum model, Chow (1988).

A Single Linear Reservoir with One Outlet

The continuity equation for a single, linear reservoir with one outlet can be written as

$$\frac{dS}{dt} = I - Q \quad (17.31)$$

where t is time, and I is the inflow rate to the reservoir.

Combining equation (17.8) and (17.9) yields a first-order, linear differential equation which can be solved explicitly

$$\frac{dQ}{dt} + \frac{1}{k}Q(t) = \frac{1}{k}I(t) \quad (17.32)$$

If the inflow (I) to the reservoir is assumed constant, the outflow (Q) at the end of a time step dt can be calculated by the following expression

$$Q(t + dt) = Q_t e^{-dt/k} + I(1 - e^{-dt/k}) \quad (17.33)$$



A Single Linear Reservoir with Two Outlets

The outflows from a linear reservoir with two outlets can also be calculated explicitly. In this case storage is merely, instead of Eq. (1) given as

$$S = k_p Q_p = k_o Q_o + thd \quad (17.34)$$

where k_p is the time constant for the percolation outlet, Q_p is percolation, k_o is the time constant for the overflow outlet, Q_o is outflow from the overflow outlet, and thd is the threshold value for the overflow outlet.

Combining equation (17.34) and (17.32) and solving for S , still assuming I is constant in time, yields the following expressions for Q_p and Q_o at time $(t+dt)$.

$$Q_p = Q_{p_i} e^{\frac{k_o + k_p}{k_p k_o} dt} + \frac{k_o}{k_p + k_o} \left(I + \frac{thd}{k_o} \right) \left(1 - e^{\frac{k_o + k_p}{k_p k_o} dt} \right) \quad (17.35)$$

$$Q_o = \frac{k_p Q_p - thd}{k_o} \quad (17.36)$$

17.2.2 General Description

In the linear reservoir method, the entire catchment is subdivided into a number of subcatchments and within each subcatchment the saturated zone is represented by a series of interdependent, shallow interflow reservoirs, plus a number of separate, deep groundwater reservoirs that contribute to stream baseflow. An example of a subdivision of a catchment area is outlined in Figure 17.8, where the topographical zones represent the interflow reservoirs in the model. If a river is present, water will be routed through the linear reservoirs as interflow and baseflow and subsequently added as lateral flow to the river. If no river is specified, the interflow and baseflow will be simply summed up and given as total outflow from the catchment area. The lateral flows to the river (i.e. interflow and baseflow) are by default routed to the river links that neighbour the model cells in the lowest topographical zone in each subcatchment.

Interflow will be added as lateral flow to river links located in the lowest interflow storage in each catchment. Similarly, baseflow is added to river links located within the baseflow storage area.

The infiltrating water from the unsaturated zone may either contribute to the baseflow or move laterally as interflow towards the stream. Hence, the

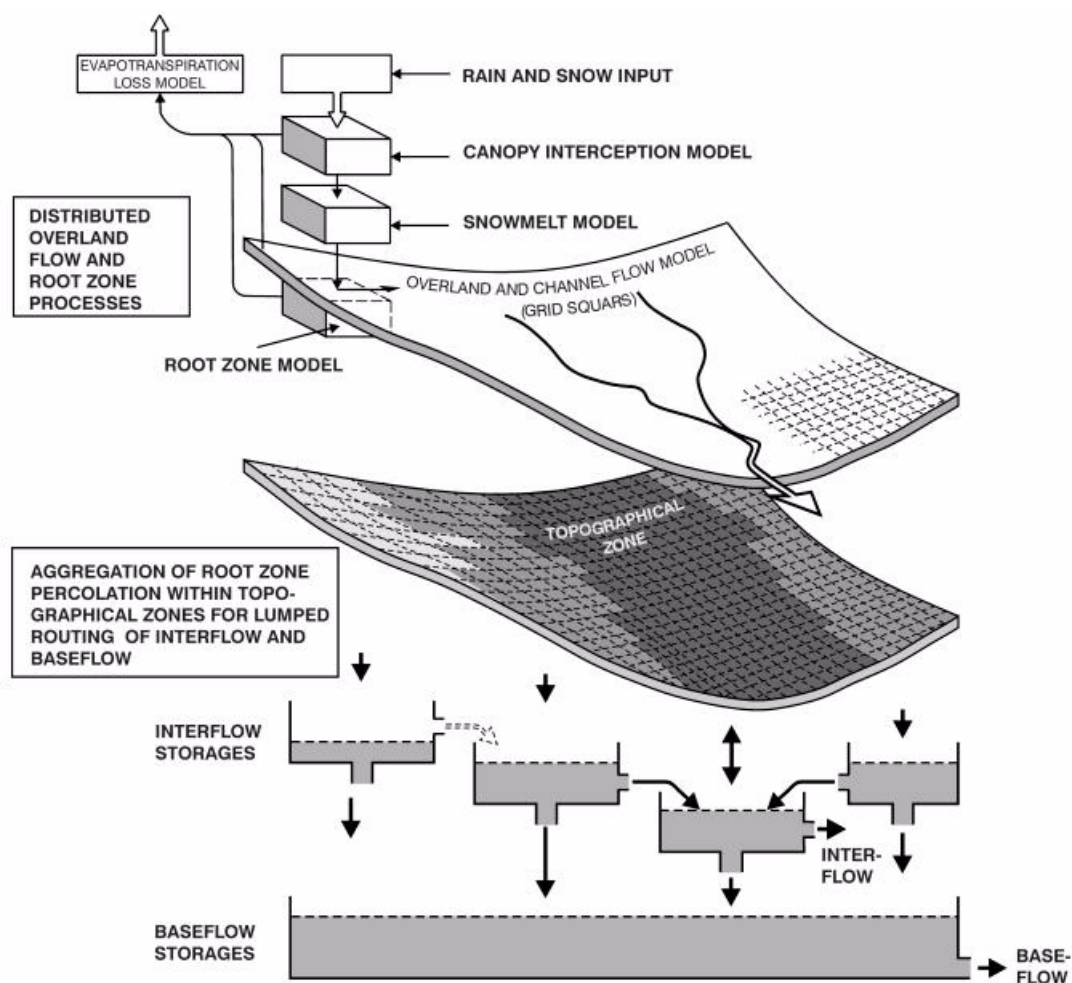


Figure 17.8 Model Structure for MIKE SHE with the linear reservoir module for the saturated zone

interflow reservoirs have two outlets, one outlet contributes to the next interflow reservoir or the river and the other contributes to the baseflow reservoirs. The baseflow reservoirs, which only have one outlet, are not interconnected.

Normally, one reservoir should be sufficient for modelling baseflow satisfactorily. However, in some cases, for example in a large catchment, hydraulic contact with a river is unlikely to be present everywhere. In this case, more than one reservoir can be specified.

In low areas adjacent to the river branches the water table may, in periods, be located above or immediately below the surface. In this case, it will contribute more to total catchment evaporation than the rest of the area. To strengthen this mechanism water held in the part of the baseflow reser-

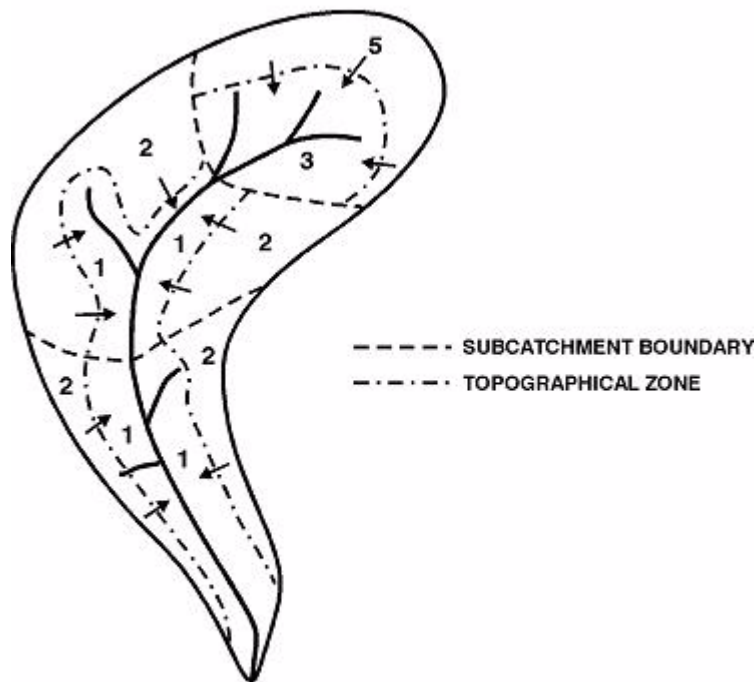


Figure 17.9 Example of Desegregation of a Catchment into Sub-catchments and Topographical Zones

voirs beneath the lowest interflow zone may be allowed to contribute to the root zone when the soil moisture is below field capacity.

Previous experience with lumped conceptual models shows that the linear reservoir approach is sufficient for an accurate simulation of the interflow and baseflow components, if the input to the reservoirs can be assessed correctly and the time constants of the outlets are known. Due to the distributed approach and physically based representation used in MIKE SHE in the overland and unsaturated zone flow components, an accurate simulation of soil moisture drainage in space and time is provided in MIKE SHE for the linear reservoir module. The time constants on the other hand are basically unknown for an ungauged catchment but a fair estimate may be obtained from an evaluation of the hydrogeologic conditions and/or from gauged catchment with similar subsurface conditions.

If the UZ feedback is not included, uncertainty of the time constants will only affect routing of the baseflow and interflow components while the total volumes of runoff will remain unchanged. If UZ feedback from the baseflow reservoir is included, some of the baseflow to the stream will be transferred to the UZ storage because of ET in the unsaturated zone.

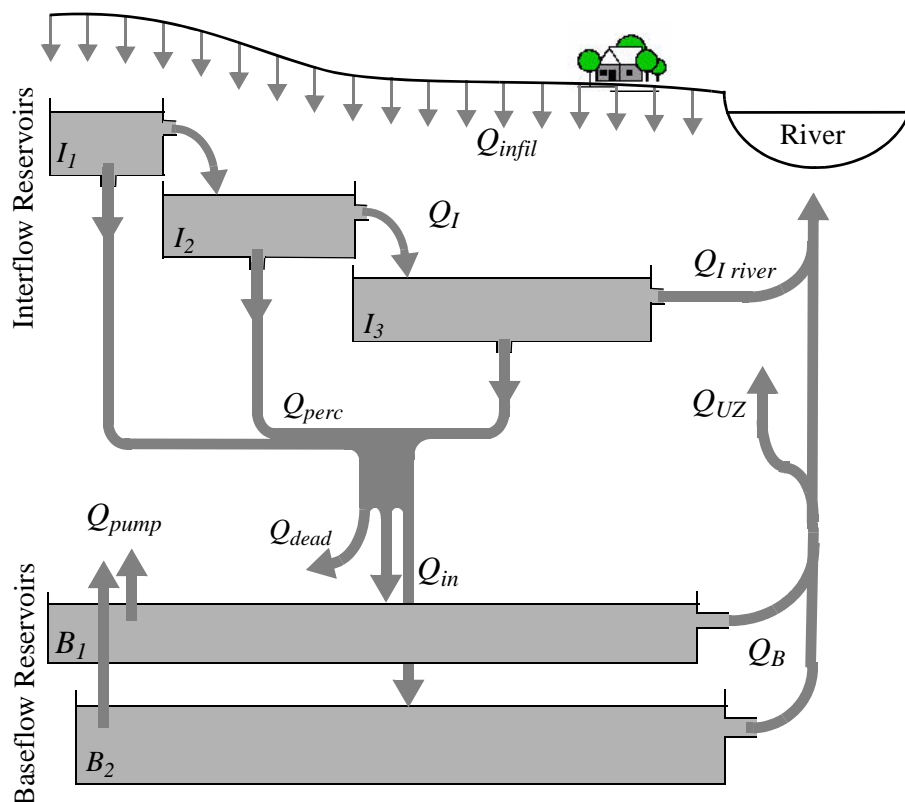


Figure 17.10 Schematic flow diagram for the Subcatchment-based, linear reservoir flow module

17.2.3 Subcatchments and Linear Reservoirs

Three Integer Grid Code maps are required for setting up the framework for the reservoirs,

- a map with the division of the model area into Subcatchments,
- a map of Interflow Reservoirs, and
- a map of Baseflow Reservoirs.

The Interflow Reservoirs are equivalent to what was called the Topographic Zones in earlier versions of the Linear Reservoir module in MIKE SHE. There is no limit on the number of subcatchments, or linear reservoirs that can be specified in the model.



The division of the model area into subcatchments can be made arbitrarily. However, the Interflow Reservoirs must be numbered in a more restricted manner. Within each subcatchment, all water flows from the reservoir with the highest grid code number to the reservoir with the next lower grid code number, until the reservoir with the lowest grid code number within the subcatchment is reached. The reservoir with the lowest grid code number will then drain to the river links located in the reservoir. If no river links are found in the reservoir, then the water will not drain to the river and a warning will be written to the run log file. For example, in Figure 17.9 Interflow Reservoir 2 always flows into Reservoir 1 and Reservoir 1 discharges to the stream network. Likewise, Reservoir 5 flows into 3, which discharges to the stream network.

For baseflow, the model area is subdivided into one or more Baseflow Reservoirs, which are not interconnected. However, each Baseflow Reservoir is further subdivided into two parallel reservoirs. The parallel reservoirs can be used to differentiate between fast and slow components of baseflow discharge and storage.

Figure 17.10 is a schematization of the flow to, from and within the system of linear reservoirs. Vertical infiltration from the unsaturated zone is distributed to the Interflow Reservoirs (Q_{infil}). Water flows between the Interflow Reservoirs sequentially (Q_I) and eventually discharges directly to the river network ($Q_{I\ river}$), or percolates vertically to the deeper Baseflow Reservoirs (Q_{perc}). The parallel Baseflow Reservoirs each receive a fraction of the percolation water (Q_{in}) and each discharges directly to the river network (Q_B). Groundwater can be removed from the Baseflow Reservoirs via groundwater pumping (Q_{pump}). If UZ feedback is included, then some of the baseflow to the stream will be added to the UZ storage (Q_{UZ}) and subsequently removed from the unsaturated zone via ET.

In some situations, the interflow reservoirs will not correspond to the areas of active baseflow in the current catchment. That is, some percolation from the interflow reservoirs may contribute to baseflow in a neighbouring watershed. This has been resolved by introducing a dead zone storage (Q_{dead}) between the Interflow and Baseflow Reservoirs

17.2.4 Calculation of Interflow

Each Interflow Reservoir is treated as A Single Linear Reservoir with Two Outlets (p. 309). Thus, from Eq. 17.30, if the water level in the linear reservoir is above the threshold water level

$$q_I = (h - h_{thresh})/k_i \quad (17.37)$$



where q_I is the specific interflow (i.e. Q/A from Eq. 17.30), h is the depth of water in the interflow reservoir, h_{thresh} is the depth of water required before interflow occurs, and k_i is the time constant for flow. If the water level is below the threshold there is no interflow.

Similarly, if there is water in the linear reservoir, specific percolation outflow can be calculated from

$$q_{perc} = h/k_p \quad (17.38)$$

where h is again the depth of water in the interflow reservoir, and k_p is the time constant for percolation flow. If the water level is at the bottom of the reservoir there is no percolation.

Combining Eqs 17.37 and 17.38 with the continuity equation

$$\frac{dh}{dt} = \frac{(q_{infil} - q_I - q_{perc})}{S_y} \quad (17.39)$$

where S_y is the specific yield, gives the following expression for h at time t when there is both q_I and q_{perc} (linear reservoir with two outlets)

$$h_t = h_{t_0} e^{\frac{-(k_i + k_p)dt}{k_i k_p S_y}} + \frac{k_i k_p}{(k_i + k_p)} \left(q_{infil} + \frac{h_{thresh}}{k_i} \right) \left(1 - e^{\frac{-(k_i + k_p)dt}{k_i k_p S_y}} \right) \quad (17.40)$$

In the case where the water level is below the threshold, the formulation for a linear reservoir with one outlet applies, which yields

$$h_t = h_{t_0} e^{\frac{-dt}{k_p S_y}} + q_{infil} \left(1 - e^{\frac{-dt}{k_p S_y}} \right) \quad (17.41)$$

Specific infiltration, q_{infil} , will normally be positive (i.e. water will be being added), but if evapotranspiration from the saturated zone is included or the net precipitation is used as input there might be a net discharge of water from the interflow reservoir. As the infiltration is a constant rate calculated explicitly in other parts of MIKE SHE, this will result in a water balance error if the interflow reservoir is empty. This will be reported in the log file at the end of the simulation.



From the level changes in the reservoir, the total average outflow can be calculated for the time step, dt . Thus, for the two outlet case,

$$q_{out} = -(h_t - h_{t0})S_y/dt + q_{infil} \quad (17.42)$$

$$q_I = \frac{k_p}{k_i + k_p}(q_{out} - h_{thresh}/k_p) \quad (17.43)$$

$$q_{perc} = q_{out} - q_I \quad (17.44)$$

and for the single outlet case (no interflow),

$$Q_{perc} = -(h_t - h_{t0})S_y/dt + Q_{infil} \quad (17.45)$$

If during a time step the reservoir level crosses one or more thresholds, an iterative procedure is used to subdivide the time step and the appropriate formulation is used for each sub-time step.

The discharge to the river, $Q_{I\ river}$, in the lowest Interflow Reservoir is simply the Q_I from that reservoir.

17.2.5 Calculation of Interflow Percolation and Dead Zone Storage

The inflow to the Baseflow Reservoirs from the Interflow Reservoirs, Q_{in} , is weighted based on the overlapping areas. Thus, if the Baseflow and Interflow reservoirs overlap, then

$$q_{in} = \sum_{i=1}^N q_{perc_i} \frac{A_{interflow_i}}{A_{baseflow}} (1.0 - DZ_{frac}) \quad (17.46)$$

where $A_{interflow}$ is the area of the Interflow Reservoir that overlaps with the Baseflow Reservoir, $A_{baseflow}$ is the area of the Baseflow Reservoir and DZ_{frac} is the fraction of the total Q_{in} that goes to the dead zone storage.

Likewise, the amount of water going to deadzone storage is given by

$$q_{dead} = \sum_{i=1}^N q_{perc_i} \frac{A_{interflow_i}}{A_{baseflow}} DZ_{frac} \quad (17.47)$$



17.2.6 Calculation of Baseflow

From Eq. 17.30, if the water level in the linear reservoir is above the threshold water level

$$q_B = \frac{(h - h_{thresh})}{k_b} \quad (17.48)$$

where h is the depth of water in the baseflow reservoir, h_{thresh} is the depth of water required before baseflow occurs, and k_b is the time constant for baseflow. If the water level is below the threshold there is no baseflow.

Similar to Eq. (17.39), for each Baseflow Reservoir

$$\frac{dh}{dt} = \frac{(q_{IN} - q_B - q_{pump})}{S_y} \quad (17.49)$$

where q_{IN} is the amount of inflow to each base flow reservoir, q_B is the amount of baseflow out of the reservoir, and q_{pump} is the amount of water removed via extraction wells from each reservoir. Both q_{IN} and q_{pump} are controlled by split fractions that distribute q_{IN} and q_{pump} between the two parallel baseflow reservoirs.

Each Baseflow Reservoir can be treated as A Single Linear Reservoir with One Outlet (p. 308). Thus, as long as the water level is above the threshold water level for the reservoir (i.e. there is still baseflow out of the reservoir),

$$h_t = h_{t_0} e^{\frac{-dt}{k_b S_y}} + (q_{IN} - q_{pump}) \left(1 - e^{\frac{-dt}{k_b S_y}} \right) \quad (17.50)$$

where k_b is the time constant for the Baseflow Reservoir.

The formula for a single outlet is applicable because there is no time constant associated with the pumping. However, Q_{pump} is also controlled by a threshold level, in this case, a minimum level below which the pump is turned off. Since this minimum level is independent of the threshold level for the reservoir itself, a case could arise, whereby there was pumping, but no baseflow from the reservoir. In this case,

$$h_t = h_{t_0} + \frac{(q_{IN} - q_{pump}) \cdot dt}{S_y} \quad (17.51)$$



If there is no pumping and no baseflow out, then the expression for the water level in the reservoir simply becomes

$$h_t = h_{t_0} + \frac{q_{IN} \cdot dt}{S_y} \quad (17.52)$$

In general, during a time step, the water level may cross one or more of the pumping or the baseflow thresholds. If this occurs, the program uses an iterative procedure to split the time step into sub-time steps and applies the appropriate formulation to each sub-time step.

17.2.7 UZ Coupling

A feedback mechanism to the unsaturated zone has been included in the module to model a redistribution of water in favour of evapotranspiration from the low wetland areas located adjacent to most rivers.

Water is redistributed from the linear reservoirs to the unsaturated zone model in the lowest topographical zone of the subcatchment, if there is a water deficit in the root zone. This deficit is called the Field Moisture Deficit (*FMD*), and is calculated as the amount of water required to bring the root zone back up to field capacity. Thus,

$$FMD = \int_0^{\text{Rootdepth}} (\theta_{FC} - \theta) dz = \sum_{i=1}^n (\theta_{FC} - \theta_i) \Delta z_i \quad (17.53)$$

where θ_{FC} is the moisture content at field capacity, θ is the moisture content in the root zone, n is the number of UZ cells in the root zone, and Δz is the height of the UZ cell.

Now, the amount of water that is available from the linear reservoirs to be redistributed to the unsaturated zone is calculated as a fraction of the baseflow:

$$S_{available} = Q_{B1} S_{y1} \cdot UZ_{frac1} \cdot dt + Q_{B2} S_{y2} \cdot UZ_{frac2} \cdot dt \quad (17.54)$$

where UZ_{frac} is a specified fraction of the baseflow that is allowed to recharge the unsaturated zone.

If $S_{available}$ is larger than or equal to the Field Moisture Deficit, then the water content of each of the root zone cells is increased to field capacity.



If $S_{available}$ is smaller than the Field Moisture Deficit, then the water content of each of the root zone cells is proportionally increased by

$$S_{feedback_i} = \theta_i + \frac{S_{available}}{FMD}(\theta_{FC} - \theta_i) \quad (17.55)$$

After the feedback calculation, the amount of baseflow to the river is reduced by the amount of water used to satisfy the Field Moisture Deficit in the unsaturated root zone, which is Q_{UZ} , in Figure 17.10.

17.2.8 Coupling to Mike 11

The discharge from the lowest Interflow Reservoir in each subcatchment is distributed evenly over the MIKE 11 river nodes located in the reservoir. Likewise, the baseflow from the Baseflow Reservoirs is distributed over the same nodes. The default distribution can be overridden by specifying specific MIKE 11 Branches and chainages.

17.2.9 Limitations of the Linear Reservoir Method

The Linear Reservoir method is a simple method for calculating overall water balances in the saturated zone. As such, it is unsuitable when a detailed spatial distribution of the water table is required. However, even given the simplicity of the method, the following simplifications and limitations should be noted:

- In the Linear Reservoir Method, the same river links are used for both of the base flow reservoirs in each of the Deep Groundwater Routing Zones. This is a limitation in the sense that occasionally you may find that fast response and the slow response baseflow may contribute to different parts of the stream.
- When calculating the unsaturated flow, the bottom boundary condition is input from a separate file - not calculated during the simulation. This means that changes in the water levels in the reservoirs will have no effect on the UZ boundary condition.
- Currently, the subcatchment-based, linear reservoir module cannot be combined with irrigation. This would require changing the way groundwater pumping was handled in the irrigation module, if the water was being extracted from a groundwater linear reservoir.



- The numerical solution used for the Linear Reservoir module assumes that the inflow to each of the linear reservoirs is constant within a time step. Strictly speaking, this is not correct as outflow from each reservoir changes exponentially during a time step. The calculation procedure uses the mean outflow from the upper reservoir as inflow to the downstream reservoir. In this way, there is no water balance error but the dynamics are somewhat dampened. If this is a problem, smaller time steps can be chosen, which will lead to a more accurate solution, as the changes in flow become smaller during each time step.
- The linear reservoir module cannot be combined with the Richards Equation description for the UZ, because the simple feedback mechanism to the UZ based on field capacity replaces the exchange with the unsaturated zone due to capillarity.





PHI Software

**TECHNICAL REFERENCE FOR
WATER QUALITY**





18 WATER QUALITY OVERVIEW

This section includes detailed descriptions of the numeric engines used for moving water in MIKE SHE, including

- Advection Dispersion - Reference (*p. 325*)
- Reactive Transport - Reference (*p. 351*)
- Particle Tracking-Reference (*p. 369*)





19 ADVECTION DISPERSION - REFERENCE

19.1 Simulation control

In the MIKE SHE water movement module, you can calculate solute transport in the different parts of the hydrological cycle. In the present version of MIKE SHE AD only three combinations are legal:

- groundwater transport can run as a stand-alone module,
- groundwater transport can be run in combination with the overland transport module, and
- all modules in combination can run.

Thus, a simulation with only overland or only the unsaturated zone is not possible and that combinations of the unsaturated zone with only the overland or groundwater component are not possible.

19.1.1 Flow Storing Requirements

The transport calculations are based on the water flow, water contents, hydraulic heads and water levels calculated in a MIKE SHE water movement simulation. Depending on the complexity of the advection/dispersion simulation, the water movement output must be stored with different storing time steps. The selected storing frequency should be sufficient to reflect the dynamics of the flow processes. However, the following two restrictions must be observed:

- The SZ head and SZ flow storing time steps must be equal, and
- The SZ storing time step must be an integer multiple of the UZ storing time step, which must be an integer multiple of the overland storing time step.

The last restriction above is controlled in the user interface.

19.1.2 Internal Boundary Conditions

If a simulation with MIKE SHE AD includes more than one part of the hydrological cycle, the solute fluxes between the different hydrologic components must be kept track of. In principle, the solute fluxes between the components follow the water flow between the components. Multiplying the flow rate with the solute concentration produces a source/sink term for the relevant components. Table 19.1 lists the solute exchange possibil-



ities between the components, in particular when one or more component is not included in the flow simulation.

Table 19.1 *Internal boundary source/sinks between hydrologic components in MIKE SHE AD*

Solutes from:	Primary solute sink:	Alternative solute sink (if primary sink unavailable):
Precipitation Fluxes	Overland flow	Unsaturated zone or Groundwater
Overland Fluxes Infiltration Overland flow “Drainage” from paved areas	Unsaturated zone MIKE 11 MIKE 11	Groundwater External boundary (none)
Unsaturated Zone Fluxes Infiltration Bypass flow	Groundwater Groundwater	External Boundary External Boundary
Groundwater Fluxes SZ Drainage Upward flux to overland Upward flux to UZ Baseflow to streams	MIKE 11 Overland flow Unsaturated Zone MIKE 11	Overland flow or External boundary (none) Overland flow (none)
MIKE 11 Baseflow to groundwater	Groundwater	(none)

An sketch of the different internal boundary conditions is shown in Figure 19.1. Each of these exchanges is detailed in the respective sub-section for each hydrologic component.

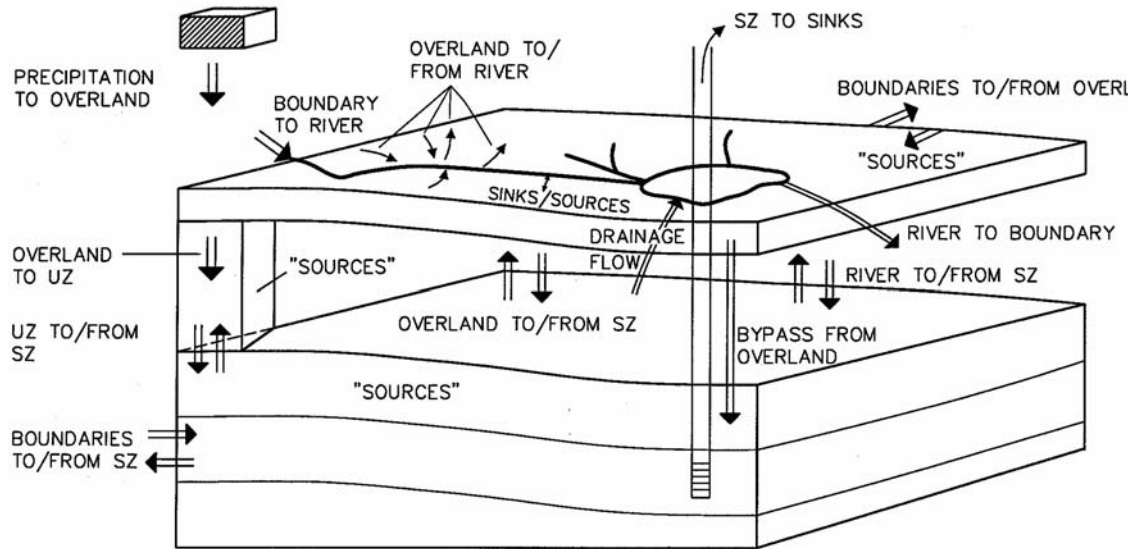


Figure 19.1 Outline of the different transport possibilities between components and boundaries.

19.1.3 Time step calculations

The time scales of the various transport processes are different. For example, the transport of solutes in a river is much faster than transport in the groundwater. The optimal time step is different for each component, where 'optimal' can be defined as the largest possible time step without introducing numerical errors. In addition, the optimal time step varies in time as a consequence of changing conditions in the hydrological regime within the catchment.

Different time steps are allowed for the different components. However, an explicit solution method is used, which sometimes requires very small time steps to avoid numerical errors. The Courant and Peclet numbers play an important role in the determination of the optimal time step.

The user can specify the maximum time step in each of the components. However, the actual simulation time step is controlled by the stability criteria, with respect to advective and dispersive transport, as well as the timing of the sources and sinks, and the simulation and storing time steps in the WM simulation.

In Figure 19.2, you can see an example of the sequence of calling each components in the MIKE SHE advection-dispersion module. The time



step in the river transport calculation is usually the smallest, whereas time step for groundwater transport is always the largest.

A transport simulation begins with the overland component followed by MIKE 11, the unsaturated zone component and the groundwater component. Figure 19.2 shows how the simulation time steps can be controlled solely by the storing time step in the flow simulation.

Solute sources and the storing of data in the different components influences the time step. For example, in Figure 19.2, a SZ source requires that all components have a break when the source starts.

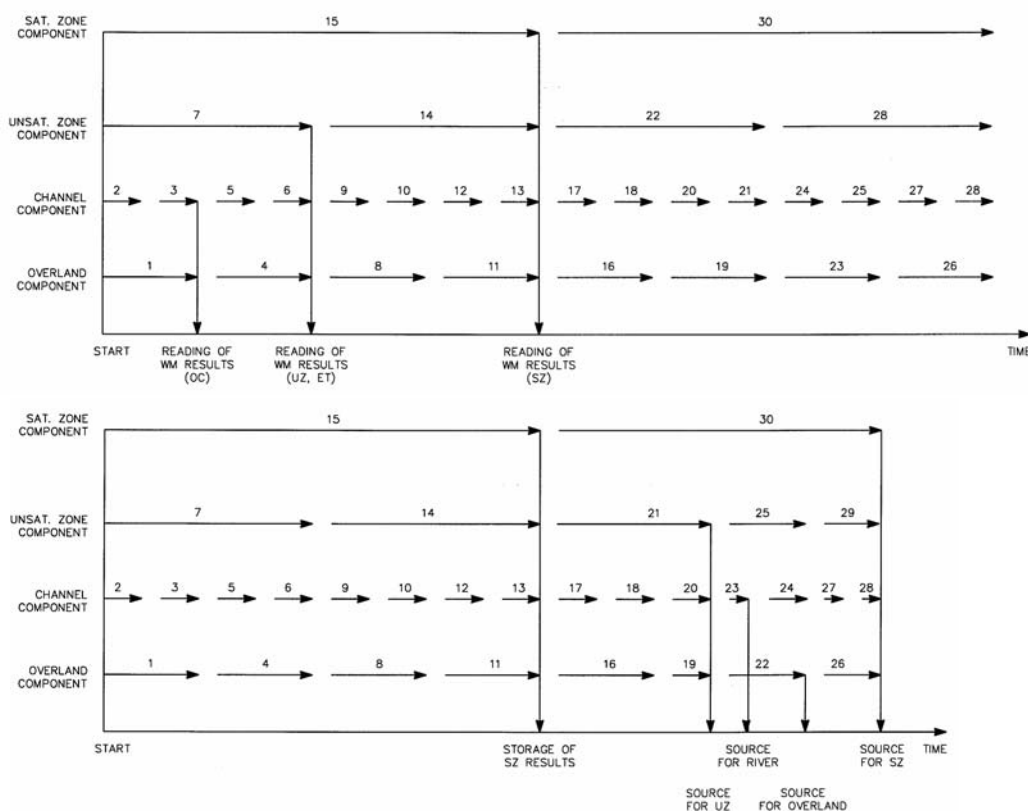


Figure 19.2 An example calculation sequence for solute transport in MIKE SHE.

Time step limitations

For each component the maximum allowable time step is determined by the advective and dispersive Courant number.



The advective Courant number in the x -direction, σ_x , is defined as:

$$\sigma_x = \frac{v_x \Delta t}{\Delta x} \quad (19.1)$$

and the dispersive Courant number, Γ_x , is defined as:

$$\Gamma = \frac{D_{11} \Delta t}{\Delta x^2} \quad (19.2)$$

The limitations are different in each flow component and will be described in more detail under the respective flow component sections.

You can also specify a maximum time step for each component, as well as a limiting solute flux per time step.

19.2 Solute Transport in the Saturated Zone

The solute transport module for the saturated zone in MIKE SHE allows you to calculate transport in 3D, 2D, layer 2D, or even 1D. However, the transport formulation is controlled by the water movement discretisation. If the vertical discretisation is uniform (except for the top and bottom layer) the transport scheme is described in a fully three-dimensional numerical formulation. If the numerical layers have different thicknesses a multi-layered 2D approach is used, where each layer exchanges flows with other layers as sources and sinks. If you specify a 1D or 2D flow simulation, the transport formulation is further simplified.

Temporal and spatial variations of the solute concentration in the soil matrix are described mathematically by the advection-dispersion equation and solved numerically by an explicit, third-order accurate solution scheme.

The forcing function for advective transport is the cell-by-cell groundwater flow, as well as groundwater head, boundary, drain and exchange flows, which are all read from the WM results files. Solute exchange between the other hydrologic components is generally simulated by means of explicit sources and sinks.



19.2.1 Governing Equations

The transport of solutes in the saturated zone is governed by the advection-dispersion equation, which for a porous medium with uniform porosity is

$$\frac{\partial c}{\partial t} = - \frac{\partial}{\partial x_i} (c v_i) + \frac{\partial}{\partial x_i} \left(D_{ij} \frac{\partial c}{\partial x_j} \right) + R_c \quad i, j = 1, 2, 3 \quad (19.3)$$

where c is the concentration of the solute, R_c is the sum of the sources and sinks, D_{ij} is the dispersion coefficient tensor and v_i is the velocity tensor.

The advective transport is determined by the water fluxes (Darcy velocities) calculated during a MIKE SHE WM simulation. To determine the groundwater velocity, the Darcy velocity is divided by the effective porosity

$$v_i = \frac{q_i}{\theta} \quad (19.4)$$

where q_i is the Darcy velocity vector and θ is the effective porosity of the medium.

The mathematical formulation of the dispersion of the solutes follows the traditional formulations generalised to three dimensions. This formula was developed assuming that the dispersion coefficient is a linear function of the mean velocity of the solutes. In the three-dimensional case of arbitrary flow-direction in an anisotropic aquifer, the dispersion tensor, D_{ij} , contains nine elements, giving a total of 36 dispersivities. The general formulation of the dispersion tensor is derived in Scheidegger (1961) and can be written as

$$D_{ij} = a_{ijmn} \frac{v_n v_m}{U} \quad (19.5)$$

where a_{ijmn} is the dispersivity of the porous medium (a fourth order tensor), v_n and v_m are the velocity components, and U is the magnitude of the velocity vector.

The derivation of D_{ij} and a_{ijmn} in MIKE SHE follows the work of Bear and Verruijt (1987). Two simplifications have been introduced with respect to dispersivity



- isotropy, and
- anisotropy with axial symmetry around the z-axis.

These simplifications are reflected in the number of non-zero dispersivities to be specified. Under isotropic conditions the dispersivity tensor, a_{ijmn} , solely depends on the longitudinal dispersivity, α_L , and the transversal dispersivity, α_T , in the following manner:

$$a_{ijmn} = \alpha_T \delta_{ij} \delta_{mn} + \frac{\alpha_L - \alpha_T}{2} (\delta_{im} \delta_{jn} + \delta_{in} \delta_{jm}) \quad (19.6)$$

where δ_{ij} is the Kronecker delta (with $\delta_{ij}=0$ for $i \neq j$ and $\delta_{ij}=1$ for $i=j$). In the Cartesian co-ordinate system applied in MIKE SHE, the velocity components in the coordinate directions are denoted V_x , V_y and V_z . Thus, we obtain the following expressions for the dispersion coefficients:

$$\begin{aligned} D_{xx} &= [\alpha_T (V_y^2 + V_z^2) + \alpha_L V_x^2] / U \\ D_{yy} &= [\alpha_T (V_x^2 + V_z^2) + \alpha_L V_y^2] / U \\ D_{zz} &= [\alpha_T (V_x^2 + V_y^2) + \alpha_L V_z^2] / U \\ D_{xy} &= (\alpha_L - \alpha_T) V_x V_y / U = D_{yx} \\ D_{xz} &= (\alpha_L - \alpha_T) V_x V_z / U = D_{zx} \\ D_{yz} &= (\alpha_L - \alpha_T) V_y V_z / U = D_{zy} \end{aligned} \quad (19.7)$$

This is the general equation for the dispersion coefficients in an isotropic medium for an arbitrary mean flow direction. If the mean flow direction coincides with one of the axis of the Cartesian coordinate system the expression for the dispersion coefficients simplifies even further (e.g. if V_y and V_z are equal to zero then D_{xy} , D_{xz} and D_{yz} will also be zero).

Under fully anisotropic conditions the dispersion coefficients depends on 36 dispersivities which is impractical to handle and estimate in practice. Thus, if we assume that the porous medium is symmetric around one of the axis, the number of non-zero dispersivities can be limited to five. This assumption is true if the medium is made up of layers normal to the axis of symmetry, which is the case for some geological deposits. Under these



conditions the following expression for the a_{ijmn} terms (Bear and Verruijt (1987)) have been derived:

$$\begin{aligned}
 a_{ijmn} = & a_I \delta_{ij} \delta_{mn} \\
 & + a_{II} (\delta_{im} \delta_{jn} + \delta_{in} \delta_{jm}) \\
 & + a_{III} (\delta_{ij} h_m h_n + \delta_{mn} h_i h_j) \\
 & + a_{IV} (\delta_{im} h_j h_n + \delta_{jm} h_i h_n + \delta_{in} h_j h_m + \delta_{jn} h_i h_m) \\
 & + a_V h_i h_j h_m h_n
 \end{aligned} \tag{19.8}$$

where a_I , a_{II} , a_{III} , a_{IV} and a_V are independent parameters and h is a unit vector directed along the axis of symmetry. In MIKE SHE AD it is assumed that the axis of symmetry always coincides with the z -axis and h becomes equal to $(0,0,1)$. Five dispersivities are then introduced

- α_{LHH} - the longitudinal dispersivity in the horizontal direction for horizontal flow
- α_{THH} - the transversal dispersivity in the horizontal direction for horizontal flow
- α_{LVV} - the longitudinal dispersivity in the vertical direction for vertical flow
- α_{TVH} - the transversal dispersivity in the vertical direction for horizontal flow
- α_{THV} - the transversal dispersivity in the horizontal direction for vertical flow



Thus, the dispersion coefficients can be written explicitly by combining Eq. (19.5) and Eq. (19.8) as follows:

$$\begin{aligned}
 D_{xx} &= \alpha_{LHH} \frac{V_x^2}{U} + \alpha_{THH} \frac{V_y^2}{U} + \alpha_{THV} \frac{V_z^2}{U} \\
 D_{yy} &= \alpha_{THH} \frac{V_x^2}{U} + \alpha_{LHH} \frac{V_y^2}{U} + \alpha_{THV} \frac{V_z^2}{U} \\
 D_{zz} &= \alpha_{TVH} \frac{V_x^2}{U} + \alpha_{TVH} \frac{V_y^2}{U} + \alpha_{LVV} \frac{V_z^2}{U} \\
 D_{xy} &= (\alpha_{LHH} - \alpha_{THH}) \frac{V_x V_z}{U} \\
 D_{xz} &= \left(\frac{\alpha_{LVV} + \alpha_{LHH}}{2} - \frac{\alpha_{TVH} + \alpha_{THV}}{2} \right) \frac{V_x V_y}{U} \\
 D_{yz} &= \left(\frac{\alpha_{LVV} + \alpha_{LHH}}{2} - \frac{\alpha_{TVH} + \alpha_{THV}}{2} \right) \frac{V_y V_z}{U}
 \end{aligned} \tag{19.9}$$

and for symmetrical reasons $D_{xy} = D_{yx}$, $D_{xz} = D_{zx}$ and $D_{yz} = D_{zy}$.

Note that Eq. (19.9) can simplify to Eq. (19.7) if $\alpha_{LHH} = \alpha_{LVV} = \alpha_L$ and $\alpha_{THH} = \alpha_{TVH} = \alpha_{THV} = \alpha_T$.

Burnett and Frind (1987) suggest that the dispersion should at least allow for the use of two transverse dispersivities - a horizontal transverse dispersivity and a vertical transverse dispersivity - to describe the difference in transverse spreading which is greater in the horizontal plane than in the vertical plane. In comparison, MIKE SHE uses all five dispersivities.

The determination of the five dispersivities is always difficult so often one has to rely on experience or on empirically derived values.

The dispersion term in the advection-dispersion equation accounts for the spreading of solutes that is not accounted for by the simulated mean flow velocities (the advection). Therefore, it is obvious that the more accurate you describe the spatial variability in the hydrogeologic regime and if the grid is sufficiently fine (i.e. the variations in the advective velocity) the smaller the dispersivities you need to apply in the model. Recent laboratory and field research have shown a relationship between the spatial variability of hydrogeologic parameters and the dispersivities. However, it is still difficult to obtain sufficient knowledge about the spatial variability of, for example the hydraulic conductivity, to determine macro dispersivities applicable for solute transport models.

19.2.2 Solution Scheme

Regular Grid

The numerical solution to the advection-dispersion equation in MIKE SHE AD is based on the QUICKEST method Leonard (1979) originally introduced this method, which was further developed by Vested et. al. (1992). It is a fully explicit scheme, which applies upstream differencing for the advection term and central differencing for the dispersion term. The equations are developed to third order and the scheme is mass conservative.

When the vertical discretisation is defined in a regular grid with uniform thickness of all layers except the upper and the lower ones the numerical scheme follows the fully three-dimensional formulation below.

Neglecting the dispersion terms and the source/sink term and assuming that the flow field satisfies the equation of continuity and varies uniformly within a grid cell the advection-dispersion equation may be written in mass conservation form as:

$$\frac{\partial c}{\partial t} + \frac{\partial}{\partial x} (v_x c) + \frac{\partial}{\partial y} (v_y c) + \frac{\partial}{\partial z} (v_z c) = 0 \quad (19.10)$$

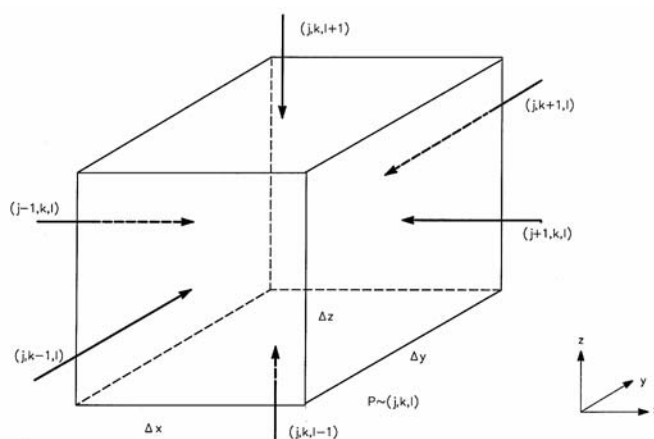


Figure 19.3 Control volume defining an internal SZ grid.



For the control volume shown in Figure 19.3 this equation is written in finite difference form as:

$$\begin{aligned}
 c_{j,k,l}^{n+1} - c_{j,k,l}^n + \sigma_x (c_{j+\gamma_x,k,l}^* - c_{j-\gamma_x,k,l}^*) \\
 + \sigma_y (c_{j,k+\gamma_y,l}^* - c_{j,k-\gamma_y,l}^*) \\
 + \sigma_z (c_{j,k,l+\gamma_z}^* - c_{j,k,l-\gamma_z}^*) = 0
 \end{aligned}
 \quad (19.11)$$

where n denotes the time index.

In Eq. (19.11) σ_x , σ_y and σ_z are the directional Courant numbers defined by

$$\sigma_x = \frac{V_x \Delta t}{\Delta x}, \quad \sigma_y = \frac{V_y \Delta t}{\Delta y}, \quad \sigma_z = \frac{V_z \Delta t}{\Delta z}
 \quad (19.12)$$

and the c^* -terms are the concentrations at the surface of the control volume at time n . As these terms are not located at nodal points they have to be interpolated from known concentration values:

$$\begin{aligned}
 c_{j+\gamma_x,k,l}^* &= \sum \delta_i c_i \\
 c_{j,k+\gamma_y,l}^* &= \sum \gamma_i c_i \\
 c_{j,k,l+\gamma_z}^* &= \sum \beta_i c_i
 \end{aligned}
 \quad (19.13)$$

The concentration c_i is the concentration around the actual point, for example $(j-1, k-1, l)$ and the weights δ_i , γ_i and β_i are determined in such a way that the scheme becomes third-order accurate. The determination of the weights is demonstrated in Vested et al. (1992) and their values are listed in Table 19.2.

A number of 8 weights has proven to be an adequate choice and their location for the determination of the $c_{j+1/2,k,l}^*$ is shown in Figure 19.4. The other “boundary” concentrations are found in a similar way



Table 19.2 Weight functions for advective transport

L_i	K_i	J_i
1 $\sigma_x \left(\frac{1}{6} \sigma_x^2 - \frac{1}{2} \sigma_x + \frac{1}{3} \right)$	$\sigma_y \left(\frac{1}{6} \sigma_y^2 - \frac{1}{2} \sigma_y + \frac{1}{3} \right)$	$\sigma_z \left(\frac{1}{6} \sigma_z^2 - \frac{1}{2} \sigma_z + \frac{1}{3} \right)$
2 $\sigma_x - 2\sigma_{xi} i=2$	$\sigma_y - 2\sigma_{yi} i=2$	$\sigma_z - 2\sigma_{zi} i=2$
3 $\sigma_x \left(-\frac{1}{6} + \frac{1}{6} \sigma_x^2 \right)$	$\sigma_y \left(-\frac{1}{6} + \frac{1}{6} \sigma_y^2 \right)$	$\sigma_z \left(-\frac{1}{6} + \frac{1}{6} \sigma_z^2 \right)$
4 $\sigma_x \left(-\frac{1}{2} \sigma_y + \frac{1}{2} \sigma_y^2 \right)$	$\sigma_y \left(-\frac{1}{2} \sigma_x + \frac{1}{2} \sigma_x^2 \right)$	$\sigma_z \left(-\frac{1}{2} \sigma_x + \frac{1}{2} \sigma_x^2 \right)$
5 $\sigma_x \left(\frac{1}{2} \sigma_x \sigma_y - \frac{1}{3} \sigma_y \sigma_z \right)$	$\sigma_y \left(\frac{1}{2} \sigma_x \sigma_y - \frac{1}{3} \sigma_x \sigma_z \right)$	$\sigma_z \left(\frac{1}{2} \sigma_x \sigma_z - \frac{1}{3} \sigma_x \sigma_y \right)$
6 $\sigma_x \left(-\frac{1}{2} \sigma_z + \frac{1}{2} \sigma_z^2 \right)$	$\sigma_y \left(-\frac{1}{2} \sigma_z + \frac{1}{2} \sigma_z^2 \right)$	$\sigma_z \left(-\frac{1}{2} \sigma_y + \frac{1}{2} \sigma_y^2 \right)$
7 $\sigma_x \left(\frac{1}{2} \sigma_x \sigma_z - \frac{1}{3} \sigma_y \sigma_z \right)$	$\sigma_y \left(\frac{1}{2} \sigma_y \sigma_z - \frac{1}{3} \sigma_x \sigma_z \right)$	$\sigma_z \left(\frac{1}{2} \sigma_y \sigma_z - \frac{1}{3} \sigma_x \sigma_y \right)$
8 $\sigma_x \left(\frac{1}{3} \sigma_y \sigma_z \right)$	$\sigma_y \left(\frac{1}{3} \sigma_x \sigma_z \right)$	$\sigma_z \left(\frac{1}{3} \sigma_x \sigma_y \right)$

The locations of the weights are determined by the points that enter into the discretisation and because the scheme is upstream centred the weights are positioned relative to the actual direction of the flow.

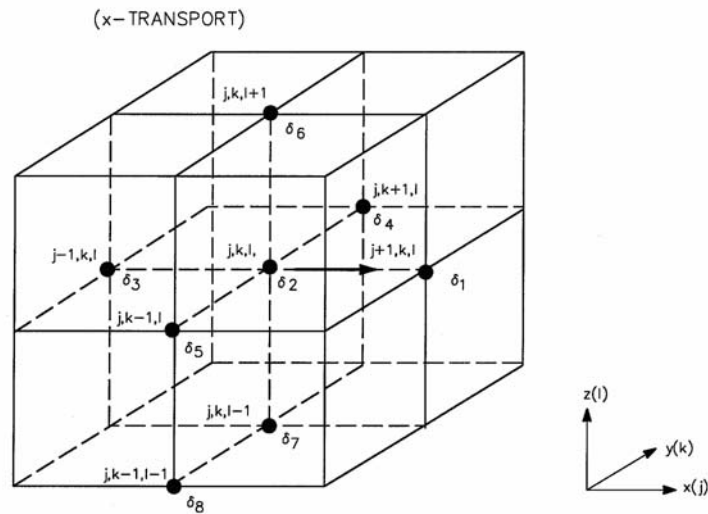


Figure 19.4 Location of interpolation weights for determination of concentrations at the location $(j+\frac{1}{2}, k, l)$ - a grid boundary.

The dispersive transport can be derived in a similar way. With the finite difference formulation of the dispersive transport components based on upstream differencing in concentrations and central differencing in dispersion coefficients the transport in the x-direction can be expressed in the following manner:

$$\begin{aligned}
 TD_{xx,j,k,l} = & -\frac{1}{2} (D_{xx,j+1,k,l} + D_{xx,j,k,l}) (c_{j+1,k,l} - c_{j,k,l}) / \Delta x^2 \\
 & -\frac{1}{2} (D_{xy,j+1,k,l} + D_{xy,j,k,l}) \cdot \\
 & (c_{j,k+1,l} + c_{j+1,k+1,l} - c_{j,k-1,l} - c_{j+1,k-1,l}) / 4\Delta y\Delta x \\
 & -\frac{1}{2} (D_{xz,j+1,k,l} + D_{xz,j,k,l}) \cdot \\
 & (c_{j,k,l+1} + c_{j+1,k,l+1} - c_{j,k,l-1} - c_{j+1,k,l-1}) / 4\Delta z\Delta x
 \end{aligned} \quad (19.14)$$

The dispersive transports in the other directions are expressed in a similar way. The dispersive transports are incorporated in the weight functions so that the mass transports can be calculated in one step.

Irregular Grid

In general the flow simulation may use varying layer thickness for the vertical discretisation of the saturated zone domain. In this case, the code checks each layer to see whether its thickness is identical with the thick-



ness of the layer above and the layer below in each of the grid cells. If this is the case this layer is handled as described for “regular grid” above (3D). If this is not the case a different approach is followed where a 2D regular grid solution based on the QUICKEST scheme is used for the horizontal transport and the vertical transport is taken into account as an explicit sink/source term.

19.2.3 Initial Conditions

The initial concentration is a fully distributed concentration field, which can be entirely uniform or constant by layer.

A boundary water flux into the model, results in a constant concentration boundary at the initial concentration value, unless explicitly specified as a time varying concentration boundary.

19.2.4 Source/Sinks, Boundary Conditions and other Exchanges

Boundary conditions for the groundwater transport component can be either

- a prescribed concentration (Dirichlet's condition), or
- prescribed flux concentration (Neumann's condition).

Catchment boundary cells with a specified head are treat as fixed concentration cells with a concentration equal to the initial concentration. A prescribed, time-varying concentration boundary can be specified at any internal node.

Prescribed flux concentration can be specified on the catchment boundary, as well as in any cell inside the model area. The flux concentration at the catchment boundary can be constant or time-varying.

Sinks can either extract pure water (concentration equal to zero) or water with the current concentration. Soil evaporation is the only sink, which removes water with a concentration equal to zero. Sinks where the concentration is equal to the actual solute concentration in the grids include pumping wells, drains and MIKE 11 nodes.

Referring back to Figure 19.1, you should note that



- If UZ transport is included, the upper boundary for the groundwater transport is the exchange of mass with the UZ component. Infiltration from (and to) the unsaturated zone is treated as a source (or sink) term. However, if the UZ flow had a shorter storing time step than the SZ flow, the concentration in the top layer of the SZ transport is updated at the UZ time step.
- If UZ bypass flow was specified, mass is transferred directly from the overland to the groundwater, with the flux equal to bypass flow multiplied with the concentration on the overland.
- Direct exchange between OL flow and SZ flow occurs when the soil is completely saturated. In this case, the infiltration from OL goes directly to the SZ and the mass flux is equal to the infiltration multiplied with the concentration on the overland.
- Exchanges with MIKE 11 are also treated explicitly as exchange flows. Inflow and outflow respectively are multiplied by the concentration in the river or the adjacent grids to the groundwater.
- SZ Drainage to the overland or MIKE 11 (or the boundary) is also treated as an SZ sink and the mass receiving component and is again calculated by multiplying the exchange flux with the concentration.

19.2.5 Transport in Fractured Media

MIKE SHE AD is able to simulate solute transport in fractured media under some simplifying conditions. If we assume that water flows only in the fractures and that solutes can enter into the soil matrix as immobile solutes the advection-dispersion equation changes to:

$$\theta_{im} \frac{\partial c_{im}}{\partial t} + \theta_m \frac{\partial c_m}{\partial t} =$$

$$- \frac{\partial}{\partial x_i} (\theta_m c_m v_i) + \frac{\partial}{\partial x_i} \left(D_{ij} \frac{\partial \theta_m c_m}{\partial x_j} \right) + R_c \quad (19.15)$$

where c is the concentration of the solute, subscripts m and im are for mobile and immobile, respectively, R_c is the sources or sinks, D_{ij} is the dispersion tensor and v_i is the velocity tensor determined from fracture porosity.



The exchange of mass between the mobile water phase in the fractures and immobile water phase is described by the traditional diffusion equation:

$$\theta_{im} \frac{\partial c_{im}}{\partial t} = \beta (c_m - c_{im}) \quad (19.16)$$

where β is the diffusion coefficient.

Diffusive exchange is included as a distributed source/sink term in the basic advection-dispersion equation.

19.3 Solute Transport in the Unsaturated Zone

The Solute Transport in the Unsaturated Zone links the transport in the overland flow and transport in the saturated zone together.

Solute transport in the unsaturated zone be simulated in both the soil matrix and macropores. Solute transport in the soil matrix is described by a 1D, unsaturated formulation of the advection-dispersion equation, which is considerably simpler than the 3D formulation in the saturated zone. Although, the unsaturated water movement calculations can be lumped together to save computational time, solute transport in the unsaturated zone is always calculated in every column. The solute transport boundary conditions and initial conditions are specified independent of any column lumping that was done in the water movement simulation.

19.3.1 Governing Equations

Soil Matrix Transport

For unsaturated solute transport in the soil matrix the advection-dispersion equation is

$$\frac{\partial c}{\partial t} = -\frac{\partial}{\partial z}(c v_z) + \frac{\partial}{\partial z}\left(D \frac{\partial c}{\partial z}\right) + R_c \quad (19.17)$$

where c is the concentration of the solute, R_c is sum of sources and sinks, D is the dispersion coefficient, and v_z is the vertical velocity.

The advective transport is determined by the water flux calculated during a MIKE SHE WM simulation. As the water flow is assumed strictly verti-



cal, this restriction applies also to the advective transport of the dissolved solutes.

To determine the velocity, v_z , the flux is divided by the moisture content:

$$v_z = \frac{q}{\theta} \quad (19.18)$$

The mathematical formulation of the dispersion of the solutes follows the formulation derived for groundwater flow with a linear relation between the dispersion coefficient and the seepage velocity but limited to one dimension. In this case the dispersion coefficient can be written as:

$$D = \alpha_L v_z \quad (19.19)$$

where α_L is the longitudinal dispersivity of the porous medium which represents the heterogeneity of the soil hydraulic parameters. α_L is allowed to vary vertically to account for different degrees of inhomogeneity in the soil. For unsaturated flow, the dispersivity is dependent on the water content, however, this relationship is neglected. .

Macro pore transport

19.3.2 Solution Scheme

Similar to the saturated zone, the unsaturated solute transport is solved explicitly, using upstream differencing for the advection term and central differencing for the dispersion term.

Neglecting the dispersion terms and the source/sink term and assuming that the flow field satisfies the equation of continuity and varies uniformly within a grid cell, the advection component is

$$\frac{\partial c}{\partial t} + \frac{\partial}{\partial z}(v_z c) = 0 \quad (19.20)$$

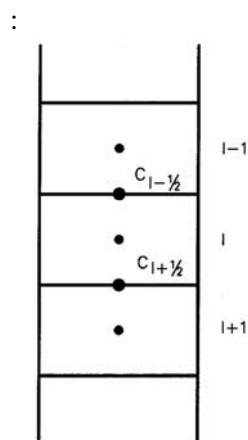


Figure 19.5 Control volume defining an internal grid.

For the control volume shown in Figure 19.5, this equation is written in finite difference form as:

$$c_I^{n+1} - c_I^n + \sigma_z (c_{I+1/2}^* - c_{I-1/2}^*) = 0 \quad (19.21)$$

where n denotes the time index, c_I is the concentration in the computational node, c^* is the interpolated concentration at the edges of the grid at time n , and σ_z is the directional Courant number defined by

$$\sigma_z = \frac{v_z \Delta t}{\Delta z} \quad (19.22)$$

As the c^* -terms are not located at the nodal points, they have to be interpolated from known concentration values. The equation for this follows the one derived for the saturated zone

$$c_{I+1/2} = \sum \beta_i c_i \quad (19.23)$$



However, in the unsaturated zone only three weights need to be determined

$$\begin{aligned}\beta_1 &= \frac{\sigma_z^2}{6} - \frac{\sigma_z}{2} + \frac{1}{3} \\ \beta_2 &= 1 - \beta_1 - \beta_3 \\ \beta_3 &= \frac{\sigma_z^2}{6} - \frac{1}{6}\end{aligned}\quad (19.24)$$

where the weights are positioned relative to the actual flow direction.

Dispersive transport can be derived in a similar way. If the finite difference formulation of the dispersive transport is based on upstream differencing in concentration and central differencing in the dispersion coefficients, the dispersive transport is

$$TD_{z,i} = -\frac{1}{2}(D_{z,i+1} + D_{z,i})(c_{i+1} - c_i)/\Delta z^2 \quad (19.25)$$

The dispersive transport is incorporated into the weights.

The above solution is strictly speaking only valid for a regular discretisation but if the resolution varies slowly the error introduced is small.

19.3.3 Initial Conditions

The initial concentration in the unsaturated zone is given as average concentrations. The unit for the concentration is [mass/volume]. This way of giving the initial conditions has the advantage that you do not have to worry about the vertical discretisation and the water content. On the other hand, you do not know the exact amount of mass introduced into the unsaturated zone, since this depends on the initial water content determined in the water movement simulation.

19.3.4 Source/Sinks, Boundary Conditions and other Exchanges

Sources for solute input into the unsaturated zone can be given in a number of ways both as point or line sources at specific depth intervals or as area sources at specific locations.

Dissolved matter can enter the unsaturated zone in three different ways. It can either be added to the precipitation as a so-called precipitation source (if the overland part is not included in the simulation) or it can be a UZ-



source introduced at a certain depth or it can enter from the saturated zone. The precipitation source option is described in the section about Overland Solute Transport.

The unsaturated zone transport component exchanges mass with the overland and the groundwater transport components as indicated in Figure 2. The transport from the overland component is a one way transport from the overland to the unsaturated zone whereas both transport to and from the groundwater can occur.

Point and line sources can be included with units of [mass/time]. Spatially distributed sources can be included with units of [mass/area/time]. In each calculation time step the solute mass in all grids nodes is updated with mass from the source.

It is not possible to introduce external sinks in the unsaturated zone. However, water can be removed by the roots or via soil evaporation, which can consequently increase solute concentrations.

19.4 Solute Transport in Overland Flow

MIKE SHE calculates the movement of solutes in overland flow, when ever ponded water exists. In surface water the mixing and spreading of solutes is mainly due to turbulence, which appears when the flow velocity exceeds a certain level. This process is known as turbulent diffusion and is generally far more important than molecular diffusion. Although this process is physically different from the spreading of solutes in groundwater, solute transport in surface water is usually still described using the advection-dispersion equation. Similar to the saturated zone, the 2D advection-dispersion equation for overland transport is solved using the explicit, third-order accurate QUICKEST scheme.

As for solute transport in the saturated zone, the dispersion coefficients depend on the spatial and temporal scale of averaging. However, dispersion in surface water depends on the homogeneity of the velocity distribution in the flow cross-section. To some extent, the dispersion depends on the mean flow velocity. However, there is no general dependence between the dispersion coefficient and the mean flow velocity. Therefore, in surface water models, the dispersion coefficient is usually specified directly. In MIKE SHE, the dispersion coefficients are assumed constant in time but may vary in space.



19.4.1 Governing Equations

The transport of solutes on the ground surface is governed by the two-dimensional advection-dispersion equation

$$\frac{\partial c}{\partial t} = - \frac{\partial}{\partial x_i} (c v_i) + \frac{\partial}{\partial x_i} \left(D_{ij} \frac{\partial c}{\partial x_j} \right) + R_c \quad i, j = 1, 2 \quad (19.26)$$

where c is the concentration of the solute, R_c is sum of sources and sinks, D_{ij} is the dispersion tensor and v_i is the velocity tensor.

The velocity of water is determined from the water flux and water depth calculated during the WM simulation.

For overland transport the longitudinal and transverse dispersion coefficients (D_L and D_T) are specified directly and the dispersion coefficients applied in Eq. (19.26) are determined as for isotropic conditions in groundwater as

$$\begin{aligned} D_{xx} &= D_{11} = D_L \frac{V_x^2}{U^2} + D_T \frac{V_y^2}{U^2} \\ D_{yy} &= D_{22} = D_T \frac{V_x^2}{U^2} + D_L \frac{V_y^2}{U^2} \\ D_{xy} &= D_{yx} = D_{12} = (D_L - D_T) \frac{V_x V_y}{U^2} \end{aligned} \quad (19.27)$$

The water depth on the ground surface varies in space and time, due to variations in topography, as well as variations in precipitation, evaporation, infiltration etc. Since evaporation can concentrate a solute beyond its solubility, a mass balance of precipitated solute is maintained, where the solute will re-dissolve if additional water becomes available. The precipitation and dissolution of the solute is controlled by its solubility.

19.4.2 Solution scheme

The solution scheme applied for overland transport uses the same QUICKEST scheme as in the saturated zone. It is a fully explicit scheme that using upstream differencing.



Neglecting the dispersion terms and the source/sink term and assuming that the flow field satisfies the equation of continuity and varies uniformly within a grid cell, the advection-dispersion equation can be written as

$$\frac{\partial c}{\partial t} + \frac{\partial}{\partial x} (v_x c) + \frac{\partial}{\partial y} (v_y c) = 0 \quad (19.28)$$

and when written in finite difference form becomes

$$c_{j,k}^{n+1} - c_{j,k}^n + \sigma_x (c_{j+\gamma_x,k}^* - c_{j-\gamma_x,k}^*) + \sigma_y (c_{j,k+\gamma_y}^* - c_{j,k-\gamma_y}^*) = 0 \quad (19.29)$$

where n denotes the time index.

In Eq. (19.29), σ_x and σ_y are the directional Courant numbers defined by

$$\sigma_x = \frac{v_x \Delta t}{\Delta x}, \quad \sigma_y = \frac{v_y \Delta t}{\Delta y} \quad (19.30)$$

and the c^* -terms are the concentrations at the surface of the control volume at time n . As these terms are not located at nodal points, they are interpolated from known concentration values by

$$\begin{aligned} c_{j+\gamma_x,k}^* &= \sum \alpha_i c_i \\ c_{j,k+\gamma_y}^* &= \sum \beta_i c_i \end{aligned} \quad (19.31)$$

The concentration c_i is the concentration around the actual point, for example $(j-1, k)$ and the weights α_i and β_i are determined in such a way that the scheme becomes third-order accurate. The determination of the



weights is demonstrated in Vested et al. (1992) and listed in Table 19.3. The other “boundary” concentrations are found in a similar way.

Table 19.3 Weight functions for advective transport

I	α	β
1	$\sigma_x(\sigma_x^2/6 - \sigma_x/2 + 1/3)$	$\sigma_y(\sigma_y^2/6 - \sigma_y/2 + 1/3)$
2	$\sigma_x - (\alpha_1 + \sigma_3 + \alpha_4 + \alpha_5)$	$\sigma_y - (\beta_1 + \beta_3 + \beta_4 + \beta_5)$
3	$\sigma_x(-1/6 + \sigma_x^2/6)$	$\sigma_y(-1/6 + \sigma_y^2/6)$
4	$\sigma_x(-\sigma_y/2 + \sigma_y^2/2)$	$\sigma_y(-\sigma_x/2 + \sigma_x^2/2)$
5	$\sigma_x(\sigma_x\sigma_y/2)$	$\sigma_y(\sigma_x\sigma_y/2)$

The locations of the weights are determined by the points that enter into the discretisation. Since the scheme is upstream centred, the weights are positioned relative to the actual direction of the flow. This is outlined in more detail for in the saturated zone Solution Scheme (p. 334) section.

The dispersive transport can be derived in a similar way. With the finite difference formulation of the dispersive transport components based on upstream differencing in concentrations and central differencing in dispersion coefficients, the transport in the x-direction can be written as

$$\begin{aligned}
 TD_{xx,j,k} = & -\frac{1}{2} (D_{xx,j+1,k} + D_{xx,j,k}) (c_{j+1,k} - c_{j,k}) / \Delta x^2 \\
 & -\frac{1}{2} (D_{xy,j+1,k} + D_{xy,j,k}) * \\
 & (c_{j,k+1} + c_{j+1,k+1} - c_{j,k-1} - c_{j+1,k-1}) / 4\Delta y\Delta x
 \end{aligned}
 \quad (19.32)$$

The dispersive transport in the y direction is done in a similar way. The dispersive transports are incorporated in the weight functions, so that the mass transports can be calculated in one step.

19.4.3 Initial Conditions

The initial concentration can be a fully distributed concentration field (e.g. measured or simulated concentrations at a certain time). The unit for the overland concentration is [mass/area].

If there is a flux of water into the model area from boundary points the flux concentration in these points will be constant in time and equal to the



initial concentration, if the flux concentration at any of these points is not specified as a time-varying source concentration.

19.4.4 Source/Sinks, Boundary Conditions and other Exchanges

Dissolved solutes can be added to the overland flow via precipitation, or from discharging groundwater. Alternatively, the solute can be added directly as a spatially distributed source on the land surface.

As indicated in Figure 19.2 the overland transport component exchanges mass with the MIKE 11, the unsaturated zone and the saturated zone. In the case of exchange to MIKE 11, the solute mass is simply added as a source term to MIKE 11. Similarly, infiltration is added as a source in the unsaturated zone. Exchange directly to the saturated zone can occur, if bypass flow is allowed, in which case, the bypass flow concentration is the same as the concentration in the overland flow. Exchange to overland flow from the saturated zone occurs if the water table rises above the ground surface.

A spatially distributed source is specified using a dfs2 file, where the source strength is given in units of [mass/area/time].

It is not possible to introduce external sinks for overland transport. However, solute concentrations can increase due to evaporation.

19.5 Solute Transport in MIKE 11

In MIKE SHE, the solute transport in the river channels is handled by the MIKE 11 Advection-Dispersion (AD) module.

In MIKE 11, the 1D advection-dispersion equation is solved using an implicit finite difference scheme that is, in principle, unconditionally stable with negligible numerical dispersion. A correction term has been added to reduce the third-order truncation error, making it possible to simulate very steep concentration gradients.

Longitudinal dispersion in channels is largely controlled by the non-uniform velocity distribution both spatially and temporally. In rivers the dispersion coefficient is normally on the order of 5 to 10 m²/s increasing to between 30 and 100 m²/s when 2D processes, such as secondary currents and wind induced turbulence begin to dominate.

MIKE 11 exchanges solutes with MIKE SHE's overland and saturated zone flow components.



Detailed information on this module is available as part of the MIKE 11 technical documentation, which can be found in .pdf form in your installation directory.





20 REACTIVE TRANSPORT - REFERENCE

Several reaction processes can be added to the solute transport calculations including

- Sorption and desorption,
- Degradation, and
- Plant uptake.

In the saturated and unsaturated all three of these processes are available. However, in the overland flow only degradation is available, but in MIKE 11 advanced reactions are possible using ECOLAB, which is a general equation solver for any kinetic reaction process.

The conservative transport of solutes in the unsaturated zone and in the groundwater is governed by the normal advection-dispersion equation described in Equation (19.3). When processes, such as sorption and decay, are included, the equation is extended to

$$\frac{\delta c}{\delta t} = - \frac{\delta}{\delta x_i} (c v_i) + \frac{\delta}{\delta x_i} \left(D_{ij} \frac{\delta c}{\delta x_j} \right) - \frac{\rho_b}{\theta} \frac{\delta c^*}{\delta t} + \left(\frac{\delta c}{\delta t} \right)_{\text{reac}} \quad (20.1)$$

where ρ_b is the bulk density of the porous medium, θ is the porosity of the porous medium, c^* is the mass of solutes sorbed per dry unit weight of solid and the term $\delta c / \delta t$ on the right hand side is a term indicating a biological or chemical reaction of the solute.

This way of describing reaction processes is very simplified and, in some cases, may give incorrect results. Nevertheless, it is a very common way of describing the reaction processes in hydrologic systems.

20.1 Sorption

Sorption includes a number of geochemical and chemical reactions, such as adsorption of solutes to the aquifer material surface by electrostatic forces (called cation exchange). If these processes occur sufficiently fast



compared with the water flow velocity they can be described by an equilibrium sorption isotherm.

Different equilibrium sorption isotherms have been identified from experimental results with different sediments, soils and rock types, see, for example, Fetter, 1993. MIKE SHE AD includes three of the most commonly applied isotherms - namely the linear, Freundlich and Langmuir equilibrium sorption isotherms.

Sorption processes that are slow compared with the water flow velocity must be described by a kinetic sorption isotherm. In MIKE SHE AD the three equilibrium sorption isotherms have been extended to include a kinetically controlled sorption process so that a certain part of the sorbed matter is “transferred” to another part of the soil material.

20.1.1 Equilibrium Sorption Isotherms

The linear sorption isotherm is mathematically the simplest isotherm and can be described as a linear relationship between the amount of solute sorbed onto the soil material and the aqueous concentration of the solute:

$$c^* = K_d c \quad (20.2)$$

where K_d is known as the distribution coefficient.

The distribution coefficient is often related to the organic matter content of the soil by an experimentally determined parameter (K_{oc}) which can be used to calculate the K_d values.

$$K_d = f_{oc} K_{oc} \quad (20.3)$$

where f_{oc} is the fraction of organic carbon.

By substituting Eq. (20.3) into Eq. (20.2), neglecting the decay term and reorganising the terms

$$\frac{\partial c}{\partial t} \left(1 + \frac{\rho_b}{\theta} K_d \right) = - \frac{\partial}{\partial x_i} (c v_i) + \frac{\partial}{\partial x_i} \left(D_{ij} \frac{\partial c}{\partial x_j} \right) \quad (20.4)$$



Commonly referred to is the retardation factor (R), which is the ratio between the average water flow velocity (v) and the average velocity of the solute plume (v_c). The retardation factor is calculated as

$$R = \frac{v}{v_c} = 1 + \frac{\rho_b}{\theta} K_d \quad (20.5)$$

The Freundlich sorption isotherm is a more general equilibrium isotherm, which can describe a non-linear relationship between the amount of solute sorbed onto the soil material and the aqueous concentration of the solute:

$$c^* = K_f c^N \quad (20.6)$$

where K_f and N are constants. The relationship between K and N is shown in Figure 20.1.

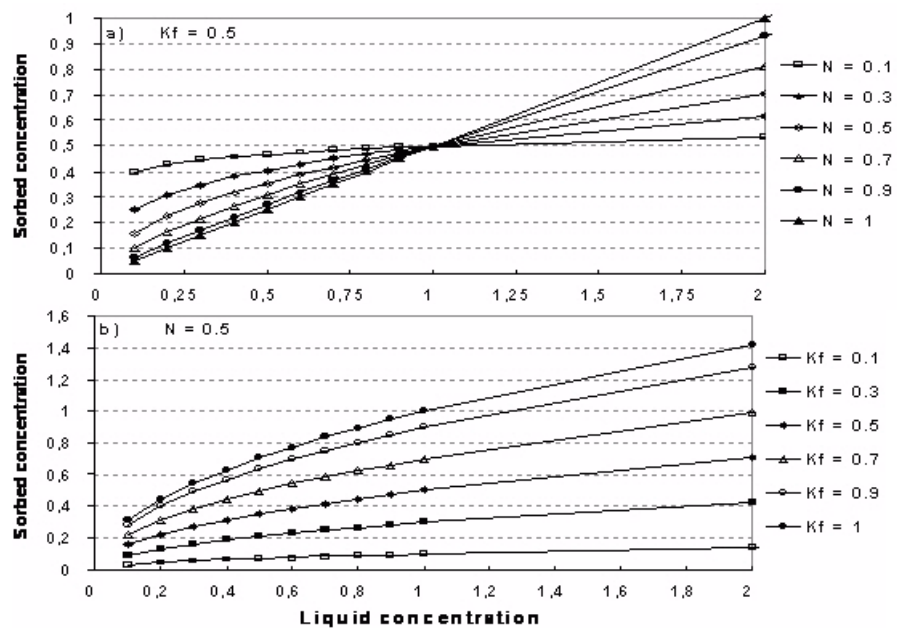


Figure 20.1 Illustration of the Freundlich isotherm. a) effect of change in N , b) effect of change in K_f

Both the linear and the Freundlich isotherm suffer from the same fundamental problem. That is, there is no upper limit to the amount of solute

that can be sorbed. In natural systems, there is a finite number of sorption sites on the soil material and, consequently, an upper limit on the amount of solute that can be sorbed. The Langmuir sorption isotherm takes this into account. When all sorption sites are filled, sorption ceases. The Langmuir isotherm is often given as

$$\frac{c}{c^*} = \frac{1}{\alpha\beta} + \frac{c}{\beta} \quad (20.7)$$

or

$$c^* = \frac{c\alpha\beta}{1 + \alpha c} \quad (20.8)$$

where α is a sorption constant related to the binding energy and β is the maximum amount of solute that can be absorbed by the soil material. The relationship between α and β is shown in Figure 20.2.

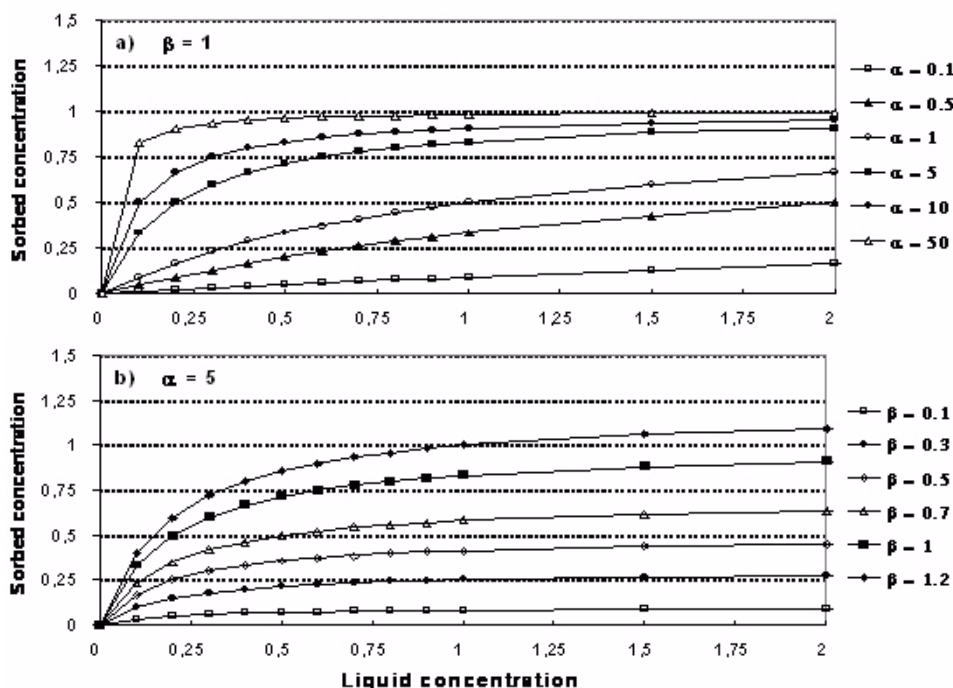


Figure 20.2 Illustration of the Langmuir isotherm. a) effect of change in α , b) effect of change in β .



20.1.2 Kinetic Sorption Isotherms

The three equilibrium sorption isotherms described so far can be extended to include kinetically controlled sorption. In the MIKE SHE AD module, a two-domain approach is used, where the sorption is assumed to be instantaneous for a fraction of the sorbed solute and rate-controlled for the remainder. A chemical formulation of this approach is given by:

$$c \leftrightarrow c^* \leftrightarrow c^{**} \quad (20.9)$$

where c^* is the amount of the solutes sorbed (described by the equilibrium sorption model), c^{**} is the amount of the sorbed matter that is converted to the kinetically controlled sorption domain. Mathematically the work of Brusseau (1995) is implemented in MIKE SHE AD as

$$\frac{\partial c^*}{\partial t} = K_I (c^{**} - c^*) \quad (20.10)$$

where K_I is the constant defining the rate of kinetic sorption. The formula is generalised so that effects of hysteresis can be taken into account, by specifying a K_I value for adsorption ($c^* > c^{**}$) and another value for desorption ($c^{**} > c^*$).

The formula is generally applicable for all the equilibrium sorption models. All constants appearing in the sorption models are assumed constant in time but may vary in space.

20.1.3 Sorption in Dual Porosity Systems

Sorption depends on the porosity and the bulk density of the soil. In dual porosity systems this is rather complicated. The distribution of sorption between the matrix and the fractures should be calculated based on the bulk density and different porosities. However, this is not always practically possible, so MIKE SHE has included a 'sorption bias factor', F_b . This allows you to explicitly control the sorption distribution between the fractures and the matrix.



Mathematically, the bulk mass available for sorption in the macro pores, ρ_{ma} , is described by:

$$\rho_{ma} = \frac{\theta_{ma} + F_b \theta_{mi}}{\theta_{ma} + \theta_{mi}} \rho_b \quad \text{for } 0 \leq F_b \leq 1$$

$$\rho_{ma} = \frac{\theta_{ma}(1 + F_b)}{\theta_{ma} + \theta_{mi}} \rho_b \quad \text{for } -1 \leq F_b < 0$$
(20.11)

where F_b is the sorption bias factor, ρ_b is the bulk mass, θ_{ma} and θ_{mi} are the macro pore and the matrix porosity respectively. The available bulk mass for sorption in the macropores is “the remainder” mass in the soil. If $F_b = 0$ the distribution of sorption sites between macro pores and matrix is assumed to be proportional to the distribution of porosities. If $F_b = 1$ sorption is assumed to occur in macro pores only and if $F_b = -1$ sorption is only occurring in the matrix region. The nature of Eq. (20.11) is illustrated in Figure 20.3.

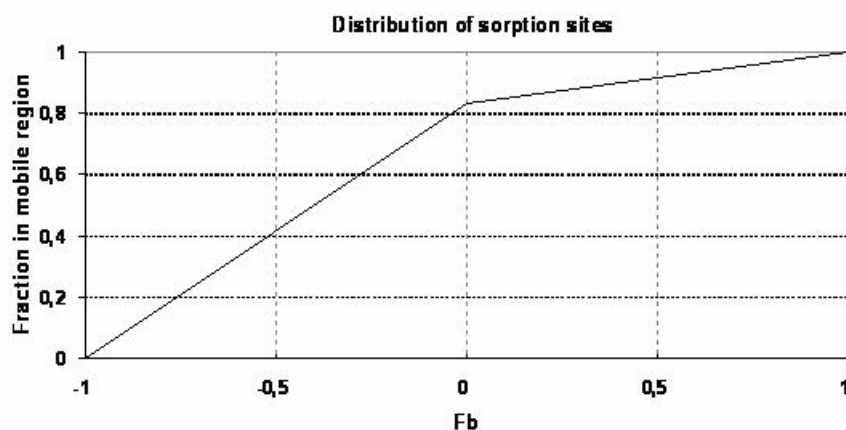


Figure 20.3 Illustration of the fraction of sorption sites located in the macro pore region (ρ_{ma}/ρ_b) as a function of F_b . Mobile porosity = 0.25, immobile porosity=0.05.

20.2 Decay

Biological degradation, radioactive decay or other kinds of attenuation of solutes can often be described as a first-order degradation process, with an



exponential decrease of concentration over a half-life. This is described in MIKE SHE as

$$\left(\frac{\partial c}{\partial t} \right)_{\text{reac.}} = \mu_{\text{ref}} c \quad (20.12)$$

where μ_{ref} is the reference degradation rate coefficient calculated by

$$\mu_{\text{ref}} = \frac{-\ln 2}{\lambda} \quad (20.13)$$

where λ is the half-life of the species.

Following the work of Boesten and van der Linden (1991), to overcome some of the difficulties in simplifying complex biological and chemical reactions, the decay in MIKE SHE is dependent of the soil moisture content and soil temperature as

$$\mu = \mu_{\text{ref}} F_w F_t \quad (20.14)$$

where the F_w is the water content function given as

$$F_w = \left(\frac{\theta}{\theta_s} \right)^B \quad (20.15)$$

and where θ is actual soil moisture, θ_s is saturated moisture content and B is an empirical constant. F_t is the soil temperature function given as:

$$\begin{aligned} F_t &= 0 && \text{for } T_s < 0^\circ\text{C} \\ F_t &= \left(\frac{T_s}{5} \right) e^{\alpha(5-T_s)} && \text{for } 0 \leq T_s \leq 5^\circ\text{C} \\ F_t &= e^{\alpha(T_s)} && \text{for } T_s > 5^\circ\text{C} \end{aligned} \quad (20.16)$$



where T_s is the actual temperature of the soil, T_{ref} is the reference temperature at which μ_{ref} is measured and α is a constant depending on T , T_{ref} , the gas constant and the molar activation.

For simplicity the soil temperature over depth is calculated as a function of the air temperature by an experimentally derived formula given by Klein (1995)

$$T_s = T_{sy} + 0.346 (T_{air} - T_{sy}) e^{(-2.7028z)} \quad (20.17)$$

where T_{sy} is the mean daily soil temperature from yesterday, T_{air} is the mean daily air temperature and z is depth.

Note that for large depths, this function responds very slowly to variations in air temperature. Therefore, long simulations may be necessary to achieve the required initial temperature distribution. An example of simulated soil temperature distributions as a function soil depth is shown in Figure 20.4.

MIKE SHE allows you to specify separate half-lives for the matrix and fractions in dual porosity models, since degradation is likely to be faster in the fractures where higher oxygen contents are more likely.

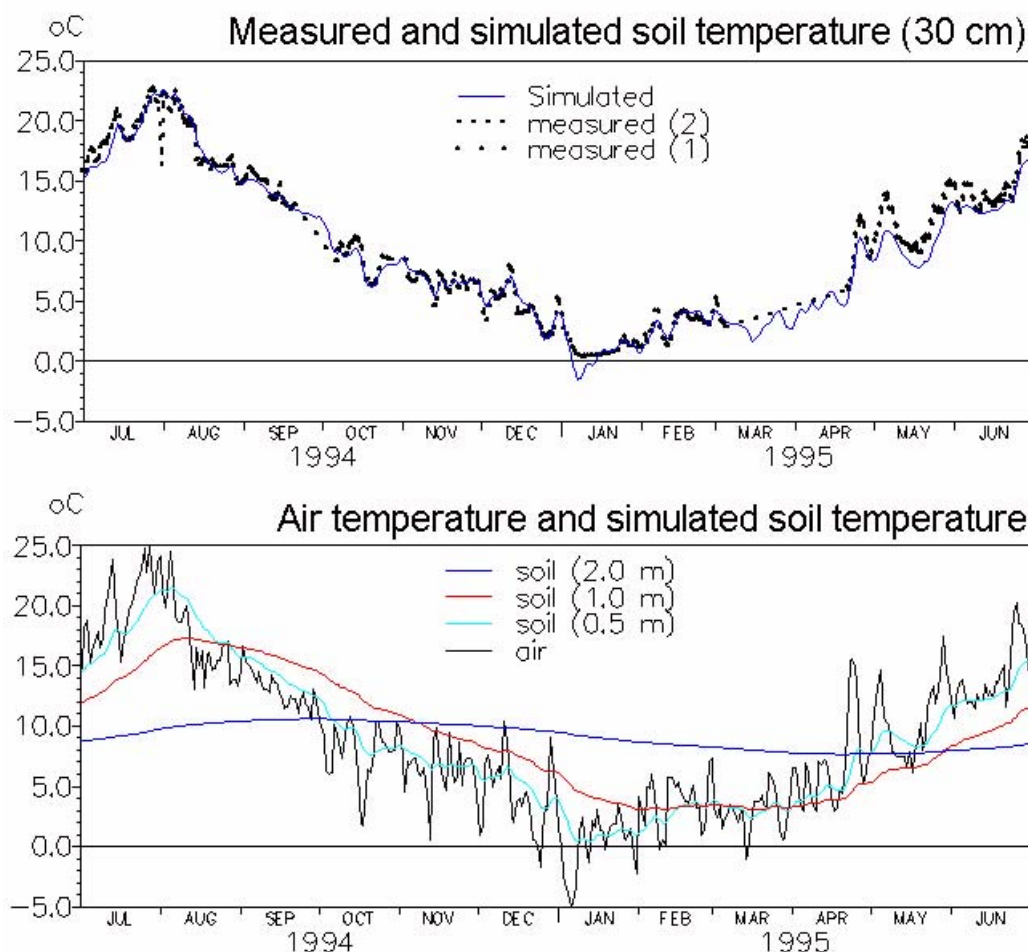


Figure 20.4 Application of the soil temperature function. Top: measured and simulated soil temperature in 30 cm depth (measurements were based on two replicates). Bottom: soil temperatures simulated in different depths based on measured air temperatures.

20.3 Plant Uptake

Plant uptake of solutes is described as passive transport, along with the transpiration stream as a function of the solute concentration in the liquid phase. Different roots have different capabilities when it comes to filtering



out various solutes. Thus, an empirical concentration factor determines to what extent the available solute is taken up by the plants.

$$R_r = f_c S_r c \quad (20.18)$$

where R_r is the sink term in the advection-dispersion equation, f_c is the concentration factor, S_r is the root water uptake and c is the liquid concentration.

20.4 Process verification

The performance of MIKE SHE's basic reactive transport module, with equilibrium and non-equilibrium sorption and degradation, has been verified against analytical solutions calculated with CXTFIT (Toride et al., 1995). The verification tests were conducted using steady-state saturated water flow through a 1 m deep column discretised in 5 cm elements. The simulations were run for one month with maximum time step equal to 15 min. Pore flow velocity was 25 cm/day, the dispersivity was 1 cm and the bulk density was 1.5 g/cm³. Furthermore, the diffusion process in fractured media with a fracture porosity of 0.25 and a matrix porosity of 0.05 is verified both without and with sorption.

The verification results confirm that the numerical solutions are satisfactory, since the calculated solute breakthrough and mass recovery curves are very similar to the analytical solutions.

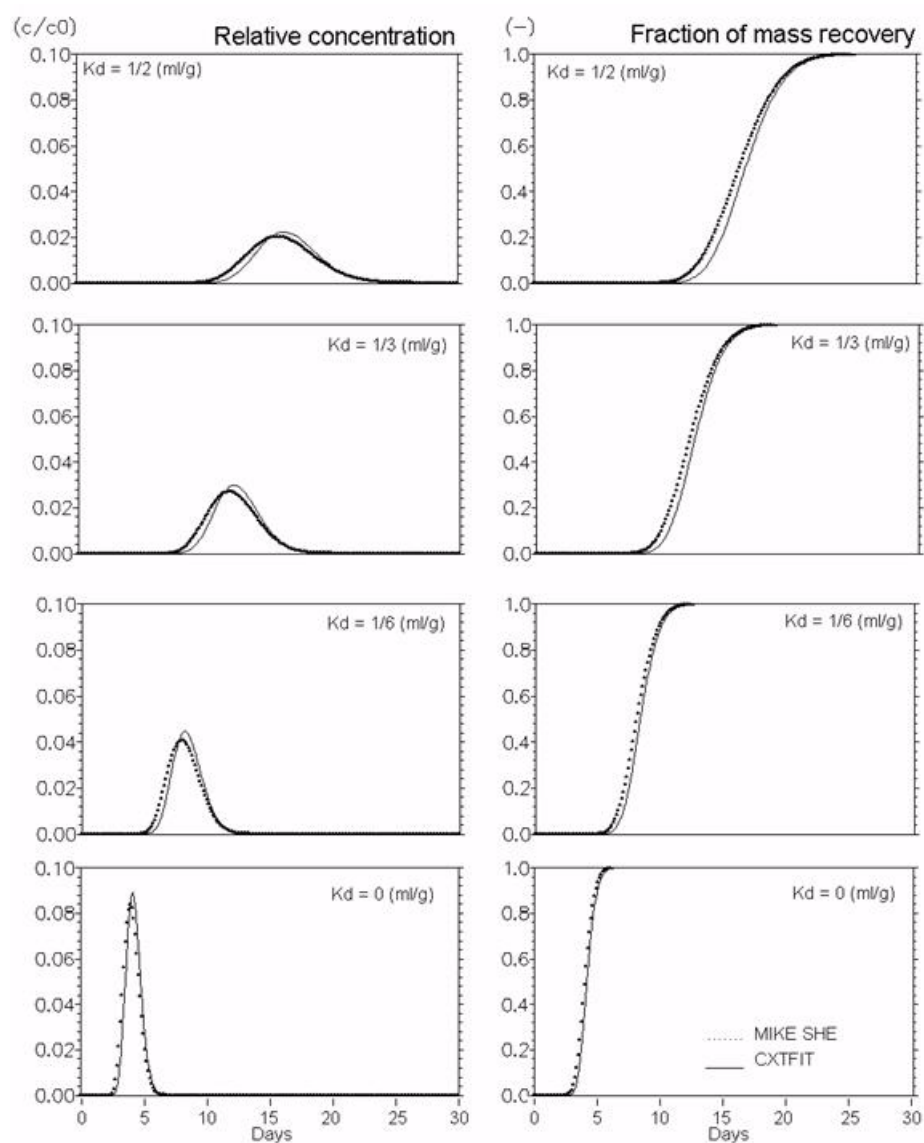


Figure 20.5 Linear equilibrium sorption. Effect of K_d (ml/g) in Eq. (20.2).

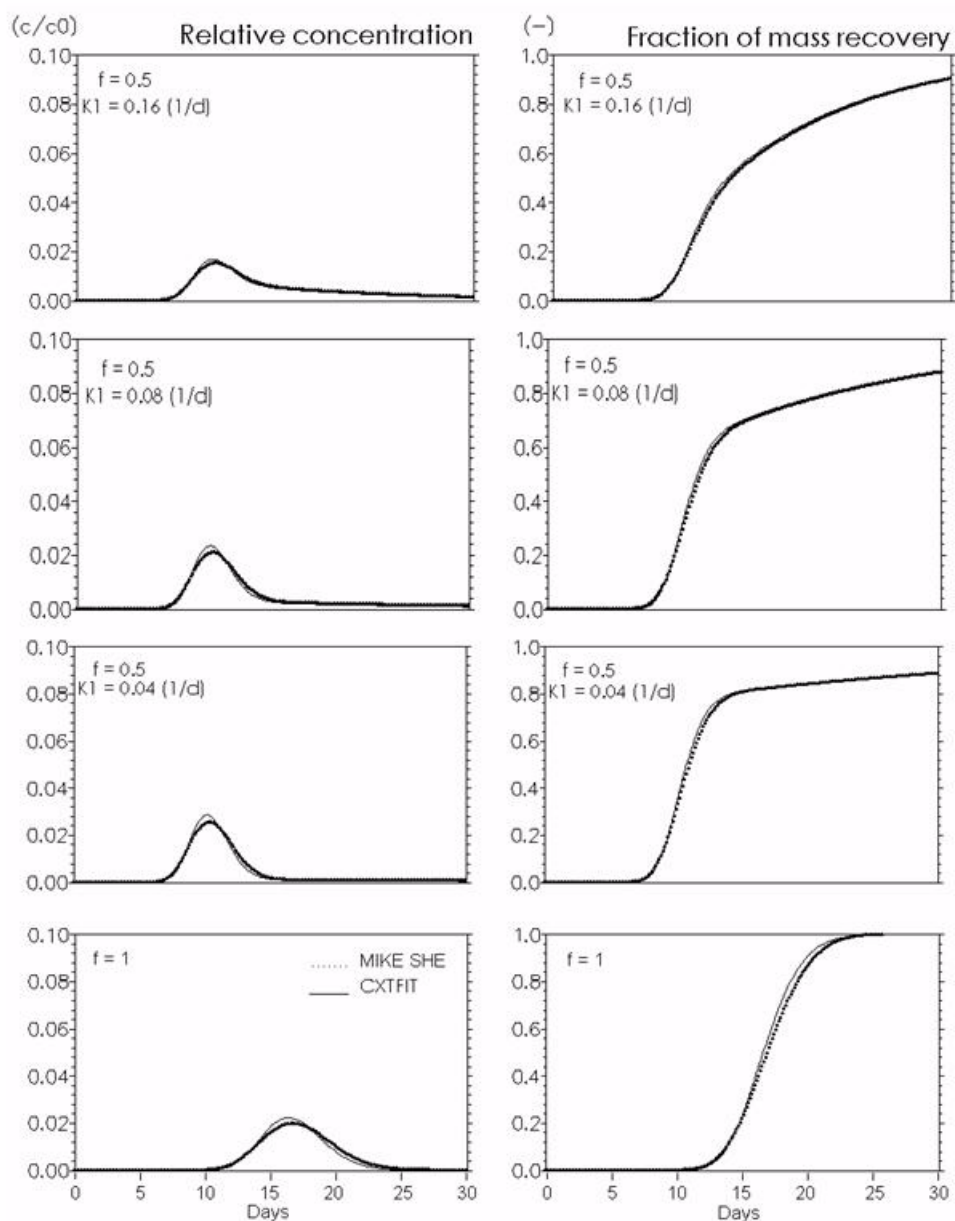


Figure 20.6 Kinetic sorption . Effects of rate constant K_1 (d^{-1}) in Eq.(20.10). Input to CXTFIT: f (fraction of equilibrium sorption sites). $K_d = 0.5$ ml/g. Input to MIKE SHE: $K_d = f \cdot K_d$ (CXTFIT). At $f=1$ the function is reduced to equilibrium sorption with $K_d = 0.5$ ml/g.

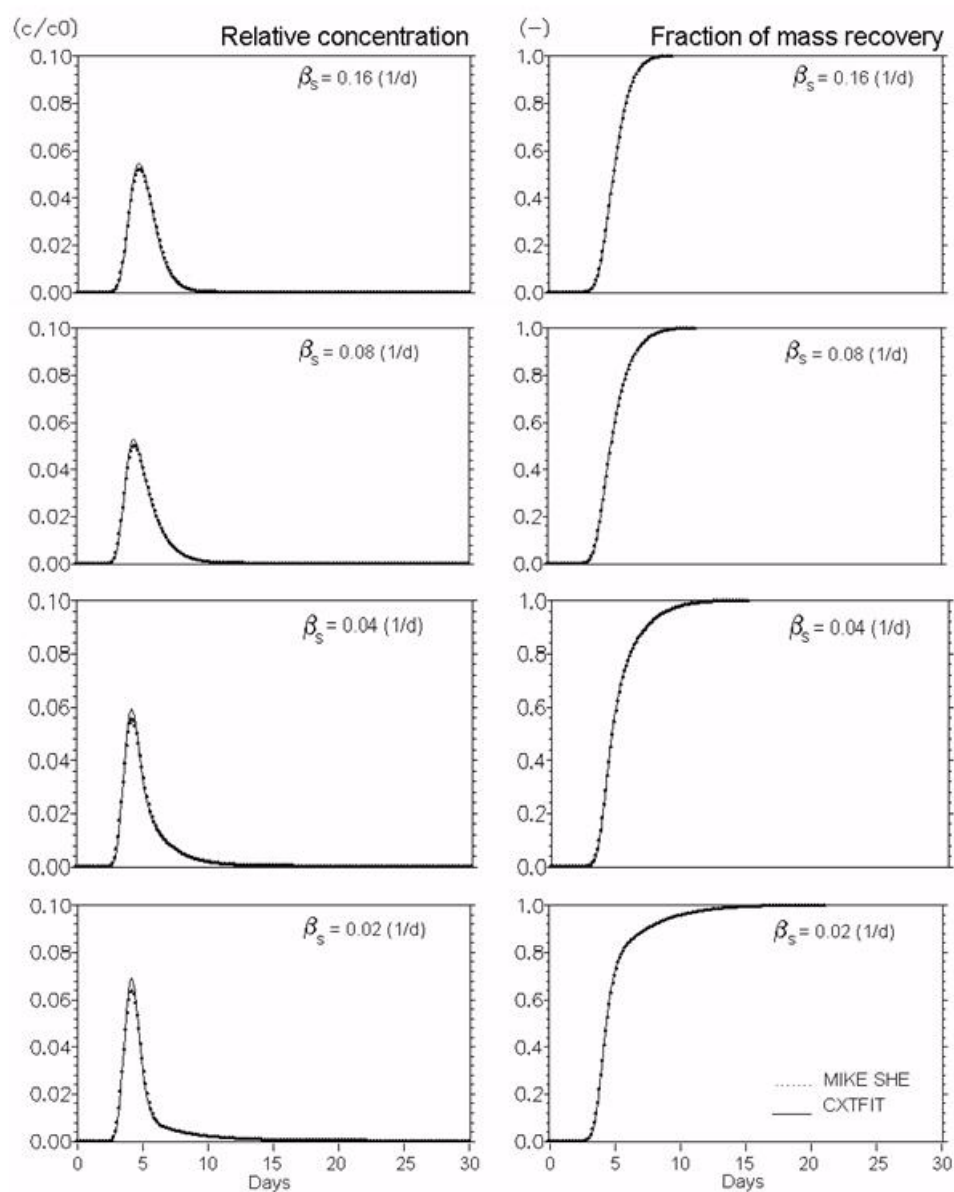


Figure 20.7 Conservative solute transport in dual porosity systems. Mobile porosity = 0.25, immobile porosity = 0.05. Effect of mass transfer coefficient (β_s in Eq. (19.16) (d^{-1})).

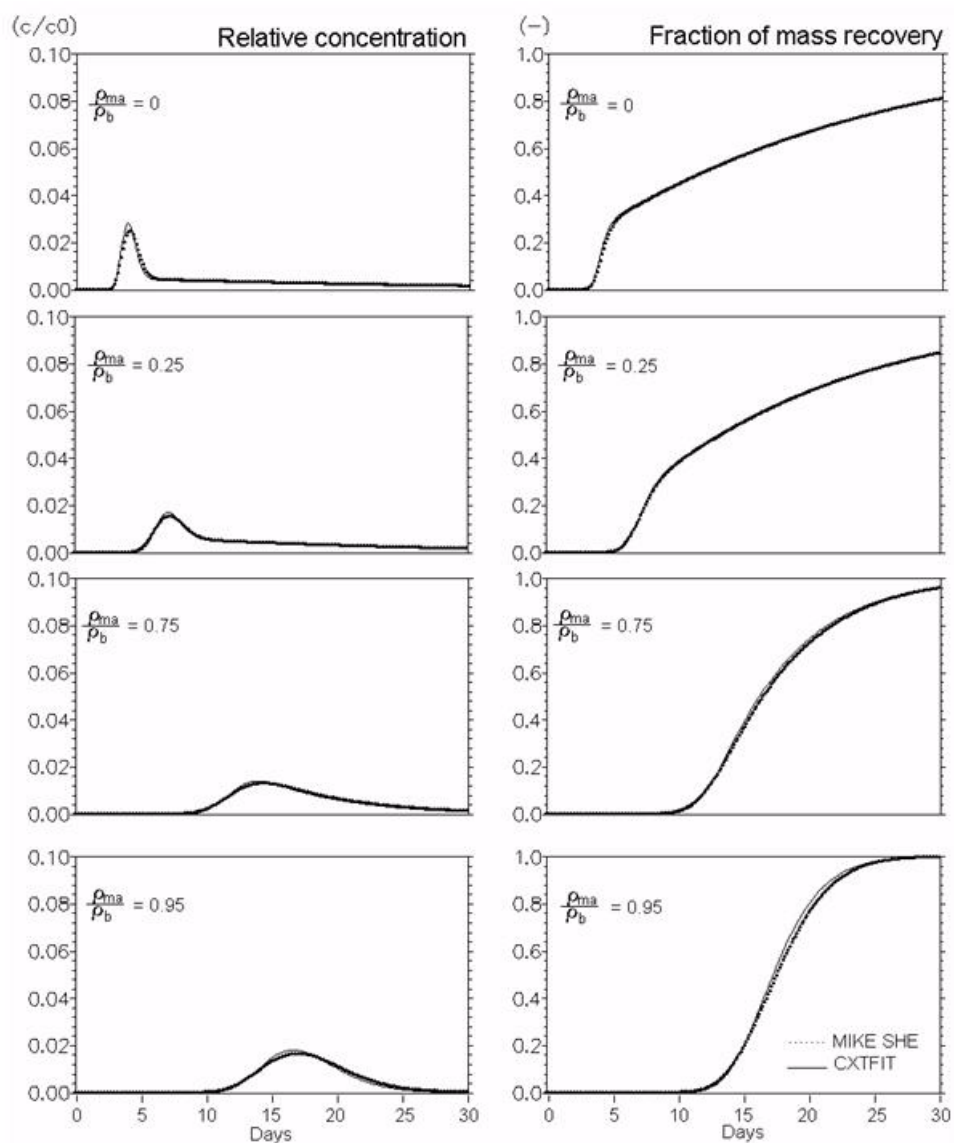


Figure 20.8 Verification of reactive solute transport in dual porosity systems. Mobile porosity = 0.25, immobile porosity = 0.05. Effect of distribution of sorption sites between mobile and immobile regions (ρ_{ma}/ρ_b Eq. 47) ($K_d = 0.5 \text{ ml/g}$, mass transfer coefficient (β_s) = 0.08 d^{-1}).

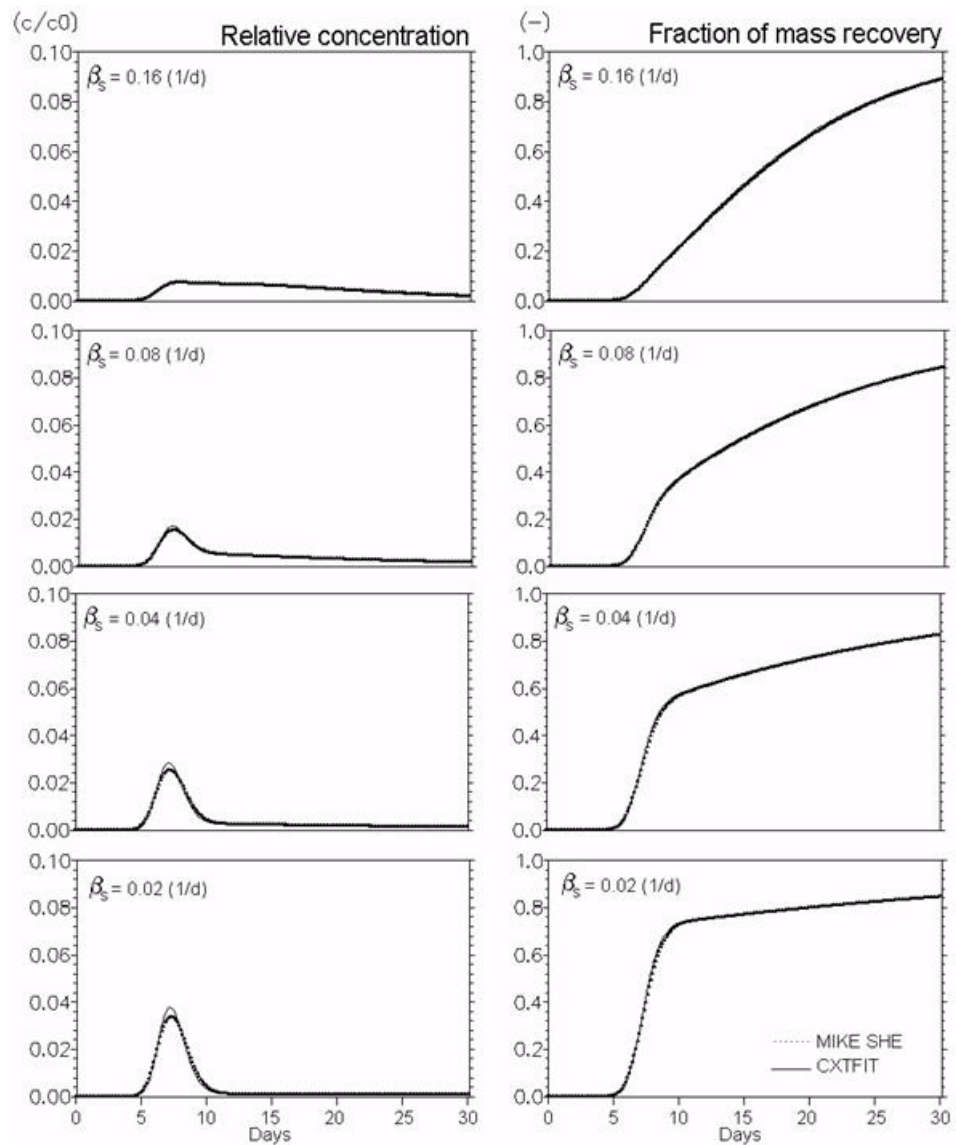


Figure 20.9 Verification of reactive solute transport in dual porosity systems. Mobile porosity = 0.25, immobile porosity = 0.05. Effect of mass transfer coefficient (β_s in Eq. (19.16) (d^{-1})). $K_d = 0.5$ ml/g, $\rho_{ma}/\rho_b = 0.25$ (Eq. (20.11)).

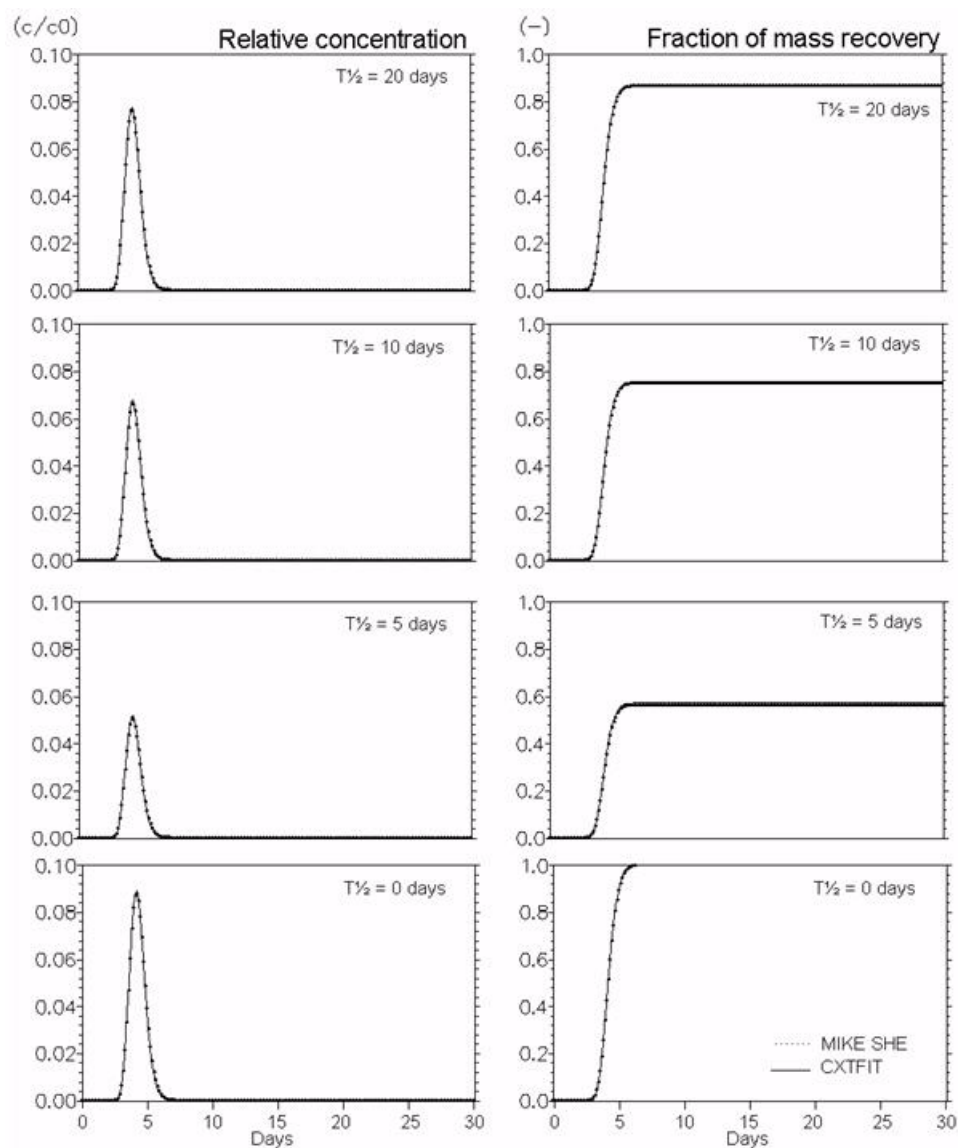


Figure 20.10 Verification of the description of first order degradation. Effect of half life time - $T_{1/2}$ (Δ) (days) (Eq. (20.13)).

20.4.1 Other Processes - Simulation Examples

Some of the process descriptions included in MIKE SHE are too complex to be verified against analytical solutions, including transport of “reactive” solutes in the unsaturated zone under different soil hydrological conditions, plant uptake of solutes, transport in macropores in the unsaturated



zone and the influence of temperature and soil moisture content on degradation processes.

To illustrate one of these cases, plant uptake of solutes is simulated in Figure 20.11. If plant uptake is included a “concentration factor” (see Eq. (20.18)) is needed, indicating to which extent solutes are taken up by the plants.

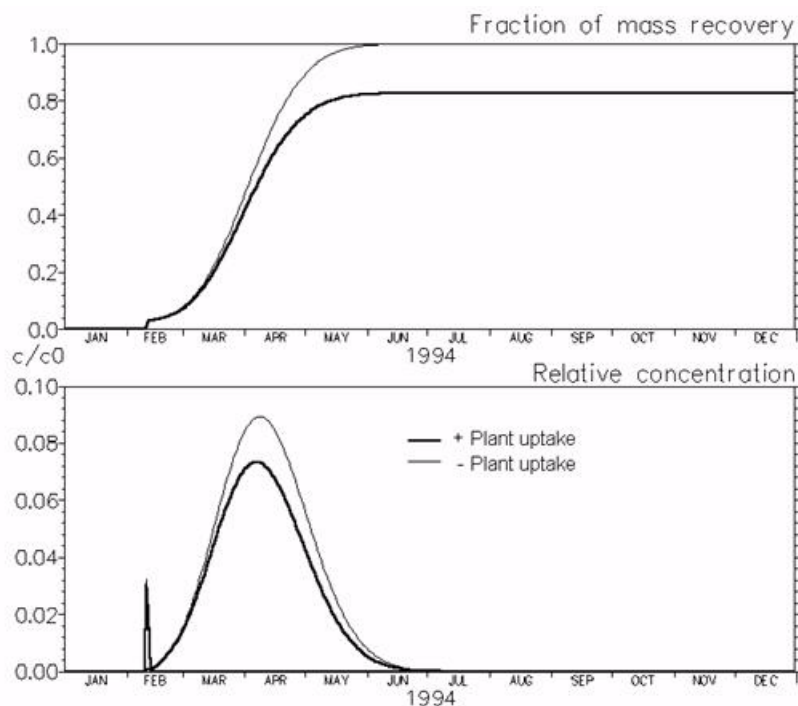


Figure 20.11 Illustration of the effect of plant uptake on solute breakthrough. Plant uptake was simulated with $f_c = 0.5$. The early peak concentrations arise from macro pore transport and were almost alike in the two simulations.





21 PARTICLE TRACKING-REFERENCE

The MIKE SHE Particle Tracking (PT) module is an alternative description of solute transport. PT calculates the location of a number of particles at every time step. The particles are displaced individually in the three-dimensional, saturated groundwater zone (SZ). The movement of each particle is composed of a deterministic part, where the particle is moved according to the local groundwater velocity calculated by the MIKE SHE, and a stochastic part, where the particle is moved randomly based on the local dispersion coefficients.

The PT module is a part of the MIKE SHE Advection Dispersion module and many of the same equations are used. For example, basically the same governing equation and the exchange of data with the MIKE SHE water movement simulation is similar in both modules. Therefore, you may find MIKE SHE AD Technical Reference a good source of additional information.

The PT module is typically used to delineate capture zones, upstream zones, and to determine groundwater age

21.1 Governing equations

The transport of solutes and particle tracking in the saturated zone is governed by the advection-dispersion equation, which for a porous medium with uniform porosity distribution is formulated as:

$$\frac{\partial c}{\partial t} + \nabla \cdot (\underline{u}c) - \nabla \cdot (\underline{D} \cdot \nabla c) = 0 \quad (21.1)$$

where c is the solute concentration, t is time, \underline{u} is the groundwater pore velocity, and \underline{D} is the dispersions tensor. In the particle model a large number of particles are moved individually in a number of time steps according to contributions from advective and dispersive transport. A particle mass is associated to each particle, which means that the number of particles in a cell corresponds to a solute concentration.

For isotropic conditions in the soil matrix the displacement of a particle is described by the following equation [Thompson et al., 1987].

$$\underline{X}_p(t_{n+1}) = \underline{X}_p(t_n) + [\underline{u}(\underline{X}_{p,n}, t_n) + \nabla \cdot \underline{D}(\underline{X}_{p,n}, t_n)]\Delta t + \underline{B}(\underline{X}_{p,n}, t_n) \cdot \underline{Z}_{p,n+1}\sqrt{\Delta t} \quad (21.2)$$



where \underline{X}_p is the particle co-ordinates, $\Delta t = t_{n+1} - t_n$ is the time step length, $\underline{Z}_{p,n+1}$ is a vector containing three independent random numbers equally distributed in the interval $[-1, +1]$ and

$$\underline{B} = \underline{R} \cdot \underline{B}^* \quad (21.3)$$

where

$$\underline{R} = \begin{bmatrix} \frac{u_x}{|\underline{u}|} & \frac{-u_y}{\beta} & \frac{-(u_y^2 + u_z^2 + u_x u_z)}{\beta |\underline{u}|} \\ \frac{u_y}{|\underline{u}|} & \frac{u_x + u_z}{\beta} & \frac{u_y(u_x - u_z)}{\beta |\underline{u}|} \\ \frac{u_z}{|\underline{u}|} & \frac{-u_y}{\beta} & \frac{u_x^2 + u_y^2 + u_x u_z}{\beta |\underline{u}|} \end{bmatrix} \quad (21.4)$$

$$|\underline{u}| = \sqrt{u_x^2 + u_y^2 + u_z^2} \quad (21.5)$$

$$\beta = \sqrt{|\underline{u}|^2 + 2u_x u_z + u_y^2} \quad (21.6)$$

$$\underline{B}^* = \begin{bmatrix} \sqrt{2(\alpha_L |\underline{u}| + D_m)} & 0 & 0 \\ 0 & \sqrt{2(\alpha_T |\underline{u}| + D_m)} & 0 \\ 0 & 0 & \sqrt{2(\alpha_T |\underline{u}| + D_m)} \end{bmatrix} \quad (21.7)$$

α_L and α_T are the longitudinal and transversal dispersion coefficients, respectively and D_m is the neutral dispersion. Using (21.2) repeatedly, the location of a particle at time $t_N = N\Delta t$ can be determined:

$$\underline{X}_p(t_N) = \underline{X}_p(t_0) + \sum_{n=0}^N ([\underline{u}(\underline{X}_{p,n}, t_n) + \nabla \cdot \underline{D}(\underline{X}_{p,n}, t_n)]\Delta t + \underline{B}(\underline{X}_{p,n}, t_n) \cdot \underline{Z}_{p,n+1} \sqrt{\Delta t}) \quad (21.8)$$



After applying (21.8) for a large number of particles, (i.e. N_p), the average solute concentration for an arbitrary volume can be calculated using (21.9)

$$c_{V,N} = \frac{1}{V} \sum_{p=1}^{N_p} m_p \delta; \delta = \begin{cases} 1; \underline{X}_{p,N} \in V \\ 0; \underline{X}_{p,N} \notin V \end{cases} \quad (21.9)$$

where m_p is the particle mass. Using this procedure an accurate solution of the advection-dispersion equation (21.1) can be obtained [Thompson et al., 1987; Thompson and Dougherty, 1988; Kitanidis, 1994].

The term $\nabla \cdot \underline{D}(\underline{X}_{p,n}, t_n)$ in (21.2) and (21.8) is assumed to be much smaller than the remaining term and is omitted for the benefit of the computational speed. This may, however in some situations result in an accumulation of particles near boundaries or stagnation points. [Kinzelbach and Uffink, 1989; Uffink, 1988; Kitanidis, 1994].

Prior to the particle tracking calculations the transient three-dimensional ground water flow field must be calculated. The groundwater velocities are used by the particle model to calculate $\underline{u}(\underline{X}_{p,n}, t_n)$ using linear interpolation for the spatial interpolation in the three directions in the grid cells. For time integration, simple Eulerian integration is used. The numerical input used by the water movement calculations is reused in the particle model as control volumes (see (21.9)) and for the specification of initial and boundary conditions.

Horizontal movement is only allowed in saturated parts of the SZ model domain. If INITSPEC -2 is used, the particles are also moved horizontally in the (fictitious) thin saturated part at the bottom of dry layers. For all other values of INITSPEC there is only vertical movement in the dry layers. The different handling of the INITSPEC -2 option was introduced to allow for a similar behaviour of PT compared to the original finite difference AD solution.

The vertical position of the particles is corrected for changes in cell thickness when a particle moves horizontally from one cell to the next. The correction uses the relative vertical location at the old location to determine the new vertical location:

$$z_{new} = \left(\frac{z_{old} - Bottom_{old}}{Top_{old} - Bottom_{old}} \right) \times (Top_{new} - Bottom_{new}) + Bottom_{new} \quad (21.10)$$



where *old* indicates the previous cell and *new* the current cell. The correction is only applied when moving horizontally from one cell to the next i.e. there is no interpolation of layer thickness during the movement within a single cell. This results in sudden changes in the vertical location at cell boundaries.

The following particle sinks can remove particles from model cells:

- constant concentration boundary receiving particles
- well
- river
- drain connected to a river or the boundary
- exchange to the unsaturated zone
- constant concentration source with a lower concentration than the calculated concentration

The following particle sources can add particles to model cells:

- constant concentration boundaries
- a solute concentration in precipitation
- a source in the saturated zone with a specified mass inflow rate
- a constant concentration source with a higher concentration than the calculated concentration

The PT module only calculates particle movements in the saturated zone. However, the volume of water removed by the wells, rivers, drains and the unsaturated zone is known. This volume of water is used to calculate the number of particles that are removed by each of the sinks using the formula:

$$n_i = n \times \frac{V_i}{V_{\text{sink}}} \times \frac{V_{\text{sink}}}{V_{\text{sink}} + V_{\text{tot}}} \quad (21.11)$$

where n_i is the number of particles removed by sink i , n is the number of particles in the saturated zone, V_i is the volume of water exchanged with sink i , V_{sink} is the volume of water exchanged with all sinks, V_{tot} is the total volume of water in the saturated zone.

Equation (21.11) is used to calculate the number of particles, which should be removed by each sink at each time step. This is, however, not necessarily a whole number of particles. The PT module takes care of this by



retaining all the fractions of particles from previous time steps until it can remove a whole particle. Particles are always assigned one by one to the sinks, with preference given to the sink in need of most particles. In case there is more than one sink in a cell, with each of these sinks requiring the same number of particles, there is a random assignment of one particle to one of these sinks. If there are any more particles left after this assignment, the next particle will then go to one of the other sinks.

Constant concentration sources and sinks at boundaries or inside the model domain are handled by calculating the number of particles that corresponds to the concentration and truncating this value to a whole number. For the mass flux source and the precipitation source the concentration is again converted to a number of particles. The whole number obtained by truncating this value is added to the compartment containing the source. The fractions that are left over after truncation are accumulated until a whole number of particles has been attained in one of the next time steps at which time an additional particle is added to the compartment in which the source is located.

Drains remove particles to rivers or to the boundary out of the model domain. Drains can also transfer particles internally in SZ. If this occurs the particles are moved from one compartment to another by the drain. Note that there is no time lag in this process.

To trace the particles, calculate transport times, capture zones, groundwater age, etc. each particle is associated with a particle identification, model time and location at which the particle was introduced in the model (time and co-ordinates of 'birth'). When particles enter sinks or are introduced into the model domain by a source this information is registered together with the source/sink type and the registration time and location before removing or adding the particle. This registration process is also used for keeping track of particles that enter registration cells. To avoid repeated registration of particles that have entered a registration cell and which are not (immediately) removed by a sink in the compartment the particle only registers the first time it enters a registration cell.





Phi Software

REFERENCES



- /1/ Abbott, M.B., Bathurst, J.C., Cunge, J.A., O'Connell, P.E., and Rasmussen, J. (1986): An introduction to the European Hydrological System - Système Hydrologique Européen, SHE, 2: Structure of a physically-based, distributed modelling system. *J. Hydrol.*, 87, 61-77.
- /2/ Abbott, M.B. and Cunge, J.A.(1982): Engineering Applications of Computational Hydraulics, Vol. I. Pitman Advanced Publ. Program, London.
- /3/ Ammentorp, H.C., A. Refsgaard (1991): Operationalizing a three-dimensional solute transport model (in Danish). Report M4/7. Landfill project. Danish Environmental Protection Agency.
- /4/ Bear, J (1979): Hydraulics of groundwater. McGraw-Hill Inc., 567 pp.
- /5/ Boesten, J.J.T.I. & van der Linden, A.M.A., 1991. Modelling the influence of sorption and transformation on pesticide leaching and persistence. *J. Environ. Qual.*, 20, 425-435.
- /6/ Brusseau, M. L., 1995: The effect of nonlinear sorption on the transformation of contaminants during transport in porous media. *Journal of Contaminant Hydrology*, 17, p 277-291.
- /7/ Bear, J. and A. Verruijt (1987): Modeling Groundwater Flow and Transport. D. Reidel Pub. Com., Dordrecht, Holland.
- /8/ Burnett, R.D. and E.O. Frind (1987): Simulation of contaminant transport in three dimensions. 2. Dimensionality Effects. *Water Resour. Res.*, 23(4), 695-705.
- /9/ Chow, T. C., Maidment, D. R. and Mays, L.W., 1988. Applied Hydrology, McGraw-Hill Series in Water Resources and Environmental Engineering, pp.201-210, 257-263
- /10/ Crawford, N.H., and Linsley, R.K., 1966, Digital simulation in hydrology: the Stanford Watershed Simulation Model IV: Technical Report no. 39, Department of Civil Engineering, Stanford University, Stanford, CA, 210p
- /11/ Ekebjerg, L. and Justesen, P. (1991): An explicit scheme for advection-diffusion modelling in two dimensions. *Computer Methods in*



- Applied Mechanics and Engineering, 88 (3), 287-297.
- /12/ FAO, 1979. Yield response to water. FAO irrigation and drainage paper no. 33. FAO, Rome Italy.
- /13/ Fetter, C.W., 1992: Contaminant Hydrogeology. Macmillan Pub. Com., New York, New York.
- /14/ Fleming G., 1975. *Computer Simulation Techniques in Hydrology*, Elsevier Environmental Science Series
- /15/ Freeze, R.A., Cherry, J.A. (1979): Groundwater, Prentice-Hall, Inc., Englewood Cliffs, N.J.
- /16/ Hill, M.C. (1990): Pre-conditioned conjugate gradient 2 (PCG2), a computer program for solving groundwater flow equations: U.S. Geological Survey Water Resources Investigations Report 90-4048, 43 p.
- /17/ Jarvis, N., 1991. MACRO - A model of water movement and solute transport in macroporous soil. Monograph, Reports and dissertations, 9. Dept. of Soil Science, Swedish Univ. of Agric. Sci., Uppsala, Sweden.
- /18/ Jensen, K.H., 1983. Simulation of water flow in the unsaturated zone including the root zone. Institute of Hydrodynamics and hydraulic engineering. Technical University of Denmark. Series Paper no. 33, 259pp.
- /19/ Jensen, K.H., H.C. Ammentorp and T. Sevel (1984): Modelling solute transport in the unsaturated zone. Nordic Hydrology 15(4/5), 223-234.
- /20/ Klein, M., 1995. PELMO - Pesticide Leaching Model, version 2.01. Users Manual. Fraunhofer-Institut für Umweltchemie und Ökotoxikologie, 57392 Schmallenberg
- /21/ Knudsen, J., 1985a. WATBAL User's Guide. Danish Hydraulic Institute, Denmark
- /22/ Knudsen, J., 1985b. WATBAL Hydrological Modelling System A Short Description. Danish Hydraulic Institute, Denmark
- /23/ Konikow, L.F. and D.B. Grove (1977): Derivation of Equations describing Solute Transport in Groundwater, US.G.S. Water Resour.



Invest. 77-19.

- /24/ Kristensen, K. J. and S.E. Jensen, 1975. A model for estimating actual evapotranspiration from potential evapotranspiration. Royal Veterinary and Agricultural University, Nordic Hydrology 6, pp. 170-188
- /25/ Leonard, B.P. (1979): A stable and accurate convective modelling procedure based on quadratic upstream interpolation. Comput. Methods Appl. Mech. Engrg., Vol. 19, pp. 59-98.
- /26/ Leonard, B.P. (1988): Simple high-accuracy resolution program for convective modelling of discontinuities, Int. Journal for Numerical Methods in Fluids, Vol. 8, pp. 1291-1318.
- /27/ McDonald, M.C., and A.W. Harbaugh (1988): A modular three-dimensional finite difference groundwater flow model: U.S. Geological Survey Techniques of Water Resources Investigations, book 6, chap.A.1, 586 p.
- /28/ Miljøstyrelsen, 1981. The surface water component of an integrated hydrological model. The Danish committee for hydrology. Suså report no. 12. The Danish Environmental Protection Agency, Copenhagen.
- /29/ Plauborg, F. and J.E. Olesen, 1991. Development and validation of the model MARKVAND for irrigation scheduling in agriculture. Tidsskrift for Planteavl's Specialserie, Beretning S 2113. Danish Institute of Plant and Soil Science. Foulum, Denmark. (In Danish)
- /30/ Refsgaard, A., Refsgaard, J.C., Clausen, T. (1993): A three-dimensional module for groundwater flow and solute transport in SHE. Danish Hydraulic Institut - Internal Note.
- /31/ Refsgaard, J.C. and J. Knudsen, 1996. Operational validation and intercomparison of different types of hydrological models, *Water Resources Research*, **32**(7): pp2189-2202
- /32/ Refsgaard, J.C., T.H. Christensen and H.C. Ammentorp (1991): A Model for oxygen transport and consumption in the unsaturated zone. *Journal of Hydrology*, 129, 349-369.
- /33/ Scheidegger, A.E. (1961): General theory of dispersion in porous media. *Jour. of Geophys. Research*, Vol. 66, no. 10.
- /34/ Thomas, R.G. (1973): Groundwater models. Irrigation and drainage.



Spec. Pap. Food Agricultural Organism. No. 21, U.N., Rome.

- /35/ Toride, N., F.J. Leij and M. Th. van Genuchten (1995). The CXT-FIT code for estimating transport parameters from laboratory or field tracer experiments. Version 2.0. Research report No. 137. U.S. Salinity Laboratory. Agricultural Research Service. USDA, Riverside, California.
- /36/ Vested, H.J., Justesen, P. and Ekebjærg, L. (1992): Advection-dispersion modelling in three dimensions. Appl. Math. Modelling, Vol. 16, pp 506-519.
- /37/ Villholth, K., 1994. Field and Numerical Investigation of Macropore Flow and Transport Processes. Series Paper no. 57. Institute of Hydrodynamics and Hydraulic Engineering, Technical University of Denmark, 230 pp.
- /38/ Yan J.J. and K.R. Smith, 1994: Simulation of Integrated Surface Water and Ground Water Systems - Model Formulation. Water Resources Bulletin, Vol. 30, No. 5, pp 1-12.





Phi Software

INDEX

**Numerics**

- 2-norm reduction-criteria 294
- 3D Finite Difference Method . . .103, 289

A

- Actual time step
 - ET 33
 - OL 33
 - UZ 33
- Air Temperature 60
- Area Inundation Exchange 239
 - Calculation - Exchange Flows . . 239
- AROOT 249
- Average steady-state river conductance 295
- Averjanov 175

B

- Baseflow 316
- Baseflow Reservoirs 106
- Bathymetry 82
- Boundary condition
 - Available 119, 121
 - Internal 121
 - Outer 118
 - Specification 118
- Boundary Conditions
 - 3D Finite Difference Method . . . 296
- Boundary conditions 218
 - Richards Equation 268
- boundary conditions
 - Dirichlet 296
 - Fourier 296
 - Neumann 296
- Bypass Flow 92

C

- Calculation
 - Baseflow 316
 - Dead Zone Storage 315
 - Exchange Flows 239
 - Interflow 313
 - Interflow Percolation 315
- Calculations
 - Lumped UZ 281

- Campbell 174
- Campbell/Burdine 176
- Canopy
 - ET 259
- Canopy Evaporation 245
- Canopy Interception 244
- Channel Flow 227
- Classification
 - Partial automatic 97
 - Specified 98
- Coefficients
 - Evapotranspiration 251
- Column Classification 91
- Computational Layers 116
- Conductivity 174
- Conjugate Gradient 43
- Control Parameters
 - OL 34
 - SZ 40
 - UZ 39
- Coupling
 - MIKE SHE and MIKE 11 228
 - Unsaturated Zone - Saturated Zone 282
- Coupling Procedure
 - Unsaturated Zone - Saturated Zone 284
- Crop Coefficient 255
- Cross-section
 - River Link 231
- Current Layer 26

D

- Database
 - Example 183
- Dead Zone Storage 315
- Detention Storage 84
- Diffusive Wave Approximation . . . 211
- Drain Codes 127
- Drain Level 125
- Drain Time Constant 126
- Drainage
 - 3D Finite Difference Method . . . 298
 - PCG Solver 299
 - SOR 306

**E****ET**

Actual	260
Canopy	259
Ponded Water	259
Saturated Zone	260
Unsaturated Zone	260
Evapotranspiration	79, 243, 258
Lakes	80
Parameters	179
Evapotranspiration Parameters	65
Exchange Flows	239
Exchange Mechanisms	
Surface Water/Aquifer	227
Extra Parameters	144

F

Finite Difference	214
Finite Difference Method	211
Flood codes	81
Flooded Area	239
Flows to MIKE SHE	232

G

Geological Layers	109
Geological Lenses	110
Geological Units	108
Gradual activation of SZ drainage	293
Gravity Flow	273
Grid series	
Output	143
Ground water table	97
Groundwater Table	101

H

Horizontal Extent	111
Hydraulic Conductivity	
Horizontal	112
Vertical	113

I

Import	139, 170
.t0 file	171
TAB delimited text file	172
Infiltration	279

Infiltration Fraction	59
-----------------------	----

Initial Conditions

Gravity Flow	275
--------------	-----

Initial conditions

3D Finite Difference Method	271
-----------------------------	-----

initial conditions

saturated zone	300
----------------	-----

Initial Potential Head	117
------------------------	-----

Interactive Map	167, 202
-----------------	----------

Interflow calculation	313
-----------------------	-----

Interflow Percolation	315
-----------------------	-----

Interflow Reservoirs	105
----------------------	-----

Irrigation

Demand	76
--------	----

Parameters	181
------------	-----

Priorities	78
------------	----

Irrigation Command Areas	69
--------------------------	----

Water Source Types	70
--------------------	----

Water Source Types - External	74
-------------------------------	----

Water Source Types - River	71
----------------------------	----

Water Source Types - Shallow Well	73
-----------------------------------	----

Water Source Types - Single Well	72
----------------------------------	----

K

Kristensen and Jensen method	244
------------------------------	-----

L

Land Use	62
----------	----

Leakage Coefficient	85
---------------------	----

Linear Reservoir

Limitations	318
-------------	-----

One Outlet	308
------------	-----

Subcatchments	312
---------------	-----

Two Outlets	309
-------------	-----

Linear Reservoir Method	102, 307
-------------------------	----------

Description	309
-------------	-----

Lower Level	111
-------------	-----

M

Manning number	83
----------------	----

Map Coordinates	201
-----------------	-----

Maximum Residual Error

PCG	293
-----	-----

SOR	305
-----	-----



- MIKE 11
- Detailed Time Series 159
 - Output 141
- MIKE 11 Water Levels 232
- Model Domain 52
- Model Grid 52
- N**
- Net Rainfall Fraction 59
- Numerical formulation
- SOR 301
- Numerical Solution 216, 263
- O**
- Option Distribution 128
- Overland Flow 82, 87, 211
- Overland Flow Routing 220
- Coupling 223
 - Directly to the river 225
 - Implementation 223
 - Theory 220
- Over-relaxation 44
- P**
- Paved Runoff Coefficient 68
- PCG Steady State Solver 294
- Plant Transpiration 246
- Ponded Water
- ET 259
- Potential flow
- PCG 290
- potential flow 290
- Precipitation 57
- Rate 58
- Pre-conditioned Conjugate Gradient 290
- Preprocessed Data 145
- Properties Editor 173
- Pumping Wells 129
- R**
- Redistribution of ponded water 224
- Relaxation Coefficient
- SOR 304
- Results 155
- Storage 135
- Results Viewer
- Gridded Data 157
- Richards Equation 262
- River Links 55
- River Overlays 25
- River-Aquifer Exchange 234
- Full Contact 234
 - Reduced Contact 235, 237
- Root Distribution 247
- Root Mass Distribution Parameter 67
- S**
- Saturated Flow 289
- Saturated Zone 102
- ET 260
- Separated Flow 87
- Shape Overlays 24
- Sheet Application Area 76
- Sheet Pile Module 300
- Simple Shape Editor 201
- Simplified ET 254
- Simplified Macropore Flow 280
- Simulation Period 28
- Simulation Specification 26
- Simulation Title 28
- Snowmelt Constants 62
- Soil Evaporation 250
- Soil Moisture 256, 276
- Soil Moisture Retention Curve 173
- Soil Profile - Definitions 93
- Soil properties
- 2-Layer UZ 99
- Sources and sinks 272
- Spatial resolution 272
- Steady-state river water depth 296
- storage 292
- Storage Coefficient 114
- Storage terms
- PCG 292
- Storing Time Steps 33
- Subcatchments 54
- Sublimation 244, 255
- Successive Overrelaxation 301
- Surface Depth
- ET 100



Surface Drainage 123

T

Time Series 156
Time series output 138
Time Step Control 30
Topography 56
Two-Layer Water Balance 275
 Simplified ET 254

U

Unsaturated Flow 261
Unsaturated Zone 90
 ET 260
Upper Level 111
UZ Classification 96
UZ Coupling 317
UZ-SZ Coupling
 Evaluation 287

V

Van Genuchten 174, 176
Vegetation 64
 Development Table 180
 Properties 64, 177
 Stages 178

W

Water Application 74
Water balance correction 216
Water Depth - Initial 85
Well Database Overlays 25
Well editor 167
Well Filters 169
Well Locations 168
Wells
 Layers 170

Y

Yield 114

